International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 10 (2021), pp. 1005-1017

Analysis of the Chassis Structure of a Formula Student Racing Car

Adrian Sebastian Fuentes Martinez[1], Albert Miyer Suarez Castrillon[2] and Sir Alexci Suarez Castrillon [3]

[1,2] Faculty of Engineering and Architecture, GIMUP, University of Pamplona, Colombia.

[3]Engineering Faculty, GRUCITE, University Francisco of Paula Santander Ocaña, Colombia.

Abstract

A structural analysis of the chassis for Formula Student single-seater cars is carried out using SolidWorks software, using alternative materials to steel, such as aluminum alloys or carbon fiber, which allow equal or greater strength, lightening the weight of the car. A quantitative and qualitative comparison of the geometry of the proposed chassis is made taking into account whether the chassis is made of steel, aluminum alloy or carbon fiber. The frontal, lateral and rollover impact is analysed, as well as the bending analysis and the braking.

Keywords: Structural analysis, SAE formula, Finite methods, Tubular chassis.

I. INTRODUCTION

The main objective that each team faces in the development of the single-seater in Formula Student, is the design of optimized geometries in each of its components and the use of composite materials such as carbon fiber, which allows to have a significant structural rigidity without sacrificing the weight of the vehicle. Among the types of chassis used, the carbon fiber single-seater has a strong and very light structure, compared to the tubular structure made of steel. However, the carbon fiber chassis has a very limited use due to the high costs of the raw material and the manufacturing process, so the most used material in the chassis by Formula SAE teams is steel. Tubular chassis offer optimum resistance to stresses at a reduced cost, but their main problem is their high weight.

The structural analysis is performed using numerical methods [1] and computer aided design simulation in such a way that the behavior of the structures can be predicted, decreasing the amount of prototypes and time in manufacturing, within the software Matlab [2]-[4] stands out for its robustness in toolbox for the process. The weight of the vehicle is very important because it can increase fuel consumption and by decreasing the unsprung masses and simulation can reduce the weight to obtain better fuel efficiency [5]. The types of loading generally analyzed in the structure are torsion, shear, lateral and vertical bending [6], and within the packages also used is Mecway [7]. The structural designs of the suspension are also analyzed, as well as the tubular elements of the structure and the material in which they are manufactured [8]. The analysis should also focus on stress situations, braking and cornering grip where Solidword and Ansy can be used for the design so that the behavior of the structure can be known [9]. Also the minimum stiffness of the roll hoops and the safety of the driver must be ensured to demonstrate the performance of the prototype [10], [11]. The relationship between torsional stiffness and weight reduction must also be calculated to reduce fuel and resources [12].

This research gathers the information about the results achieved in each of the simulations. An ideal case of torsional stiffness was proposed and a comparison was made between this and the values obtained for the different chassis alternatives. For this, the results of displacement and maximum stress of each structure were tabulated, which were used to calculate the angle of rotation which is necessary to obtain the torsional stiffness.

2. METHODOLOGY

A design of each base prototype is made with 3 materials: Steel 4130 tubular chassis, Aluminum 6061 T6 tubular chassis and Carbon Fiber Single-seater. Once the design of each base prototype is completed, we proceed to perform the various studies of simulations to validate the structure and determine the critical points of the chassis, the analysis were:

2.1 Tubular Chassis

Frontal Impact

The frontal impact test is carried out as indicated in the regulations. A force of 120 kN was applied in the direction normal to the frontal protection plane, which is distributed in the 4 nodes of the frontal protection plane, for a total of 30 kN in each one. The fixation points of the structure were the lower nodes of the main arch and the front arch. Figure 1 illustrates the pre-process analysis.

Side Impact

The side impact test is carried out in order to ensure that the structure can withstand the force transmitted due to a collision between two single-seaters or by any element of track protection in case the driver loses control of the vehicle. For this analysis the regulations indicate a force of 7 kN distributed along the side impact protection. The attachment points of the chassis are the lower nodes of the main arch and to the front arch (Figure 2).

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 10 (2021), pp. 1005-1017

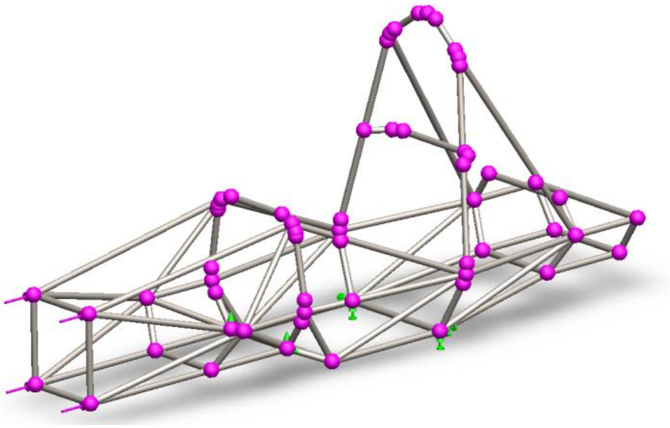

Fig. 1. Applied force and attachment points for frontal impact analysis.

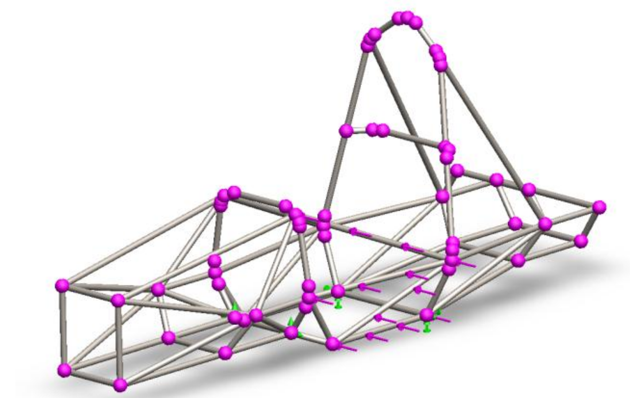

Fig. 2. Applied force and attachment points for side impact analysis.

Overturning in main arch

This test is designed to ensure that the main arch structure does not fail at any time in the event of a rollover accident. In this way the structure absorbs the impact generated, preventing the rider's head from hitting the ground, putting the rider's safety at risk. To perform the analysis the regulations indicate that a force applied at the highest point of the main arch in the directions Fx = 6 kN, Fy = 5 kN and Fz = -9 kN, should not generate a deformation greater than 25 mm avoiding the failure of the structure at any of its points. The attachment points of the chassis are the lower nodes of the main arch and the front arch. Figure 3 illustrates the pre-process of the analysis.

Front arc rollover

The analysis of the frontal arch is designed to ensure that the chassis structure can withstand the load produced by a rollover accident, this in order to ensure the pilot's body at all times. The analysis was performed in its entirety as indicated by the regulations, the load was applied at the highest point of the front arch which was in the directions Fx = 6 kN, Fy = 5 kN and Fz = -9 kN, should not generate a deformation greater than 25 mm avoiding the failure of the structure at any of its points. The attachment points of the chassis are the lower nodes of the main arch and the front arch. Figure 4 illustrates the pre-process of the analysis.

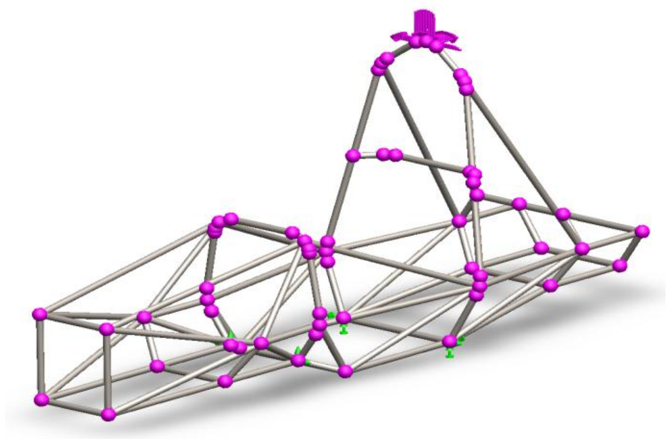

Fig. 3. Applied force and attachment points for the overturning analysis on the main arch.

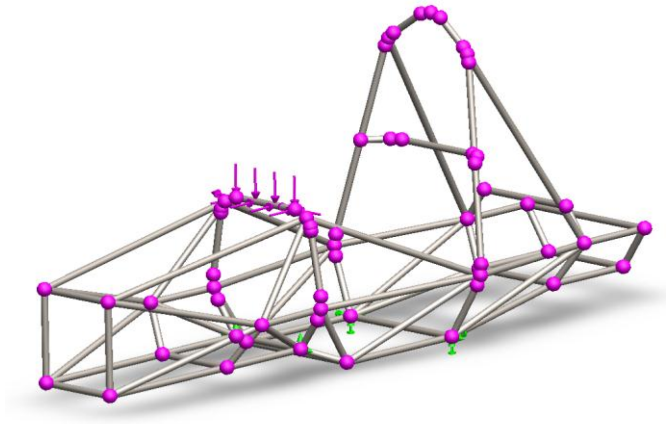

Fig. 4. Applied force and attachment points for overturning analysis in the frontal arch

Flexural Analysis

The purpose of the bending analysis is to determine the resistance of the structure against the loads produced by the mechanical components of the vehicle and the weight of the driver in static condition. The analysis is performed by simulating the loads produced by the heaviest components of the single-seater along its longitudinal axis, which are distributed in each of the anchor points of the chassis. The calculation of each force is carried out using equation 1.

$$\boldsymbol{F_{PA}} = \frac{P * g * a}{n} \quad (1)$$

Where F_{PA} is the force produced at each anchor point, P is the weight of each component, g is the acceleration of gravity, a is the acceleration suffered by the chassis in g-force, which is

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 10 (2021), pp. 1005-1017

taken as 1 when the vehicle is in static condition and n is the number of anchor points that support each mechanical component. Table 1 shows the forces produced in the bending analysis.

Table 1. Forces produced in the bending analysis

FLEXURAL ANALYSIS				
Charges	**Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg**	**Anchor points**	**Force [N] Force [N] Force [N] Force [N] Force [N] Force [N] Force [N**	**Force at each anchor point [N].**
Pilot	80	6	784.8	130.8
Engine	60	8	588.6	73.575
Suspension	30	16	294.3	18.394
Tank	18	4	176.58	44.15
Battery	15	4	147.15	36.788

At the moment of the simulation, the areas where the maximum stress of the structure is produced will be studied, through the Von Mises criterion and the points with the most critical deformation, in order to later take each one as a reference parameter in the structural optimization process. To fix the structure and thus avoid its displacement both vertically and horizontally, the four anchor points of each suspension are taken, the pre-process of the simulation is shown in Figure. 5.

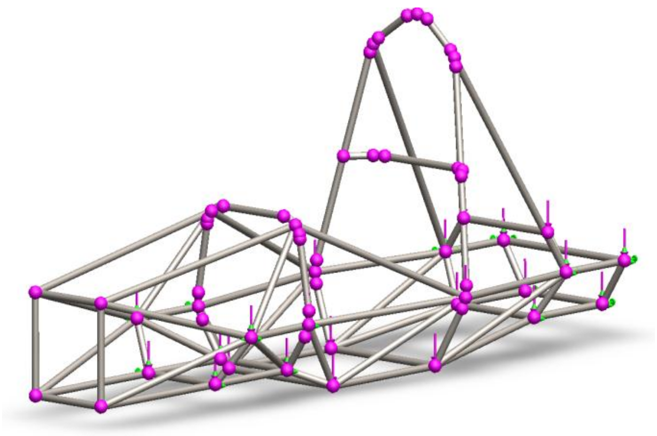

Fig. 5. Applied forces and attachment points for torsional analysis

Acceleration

When the car starts, inertia forces are produced in the entire chassis structure, which are transferred from the front suspension to the rear suspension. In the acceleration analysis the g-force is produced by the acceleration of the engine, which is provided by the engine manufacturer. The Honda CBR 660 engine has an acceleration of 8.7 m/s^2 (Honda, 2017), which equals 0.89 g. Table 2 shows the forces produced in the chassis due to the main mechanical components and the rider at the moment the vehicle accelerates.

Table 2. Forces produced in the acceleration analysis

ACCELERATION				
Charges	**Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg]**	**Anchor Points**	**Force [N] Force [N] Force [N] Force [N] Force [N] Force [N] Force [N**	**Force at each anchor point [N].**
Pilot	80	6	698.472	116.412
Engine	60	8	523.854	65.482
Suspension	30	16	261.927	16.370
Tank	18	4	157.156	39.289
Battery	15	4	130.964	32.741

In the simulation study, each of the forces will go in the opposite direction to the movement of the vehicle and are located at the anchor points of the elements. Since the axles of the suspension are responsible for transmitting the force produced by the acceleration, the structure will be fixed at the anchor points of the system avoiding the displacement in the X and Y axes. Next, the pre-process of the analysis is illustrated (Figure 6).

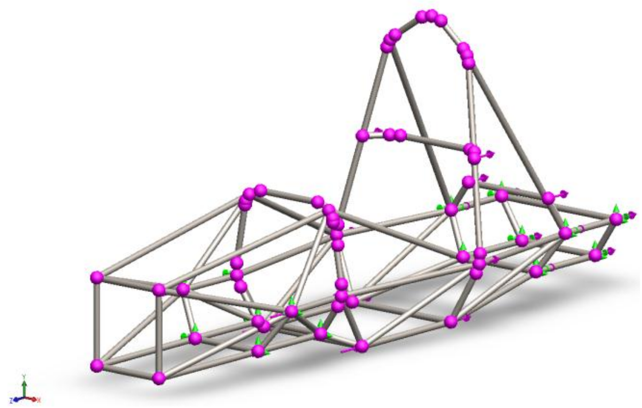

Fig. 6. Applied forces and attachment points for the acceleration analysis

Braking

The purpose of the analysis is to study the behaviour of the vehicle under the loads produced by braking at high speeds. For this, the regulation contemplates a test where the vehicle must be able to stop in the shortest possible time once 100 km/h are reached in a 25 m straight trajectory. The calculation of this deceleration is done using the criteria of body dynamics (Equation 2),

$$\boldsymbol{a} = \frac{v_2^2 - v_1^2}{2 * (x_1 - x_2)} \tag{2}$$

Wherein v_2 y v_1 are the initial and final velocity respectively in m/s^2, x_1 y x_2 are the initial and final position of the vehicle in m. Once the calculation was done it was determined that the value of the acceleration for the study is $15.432\ m/s^2$which is equal to $1.57g$. Table 9 shows the forces produced for this study.

Table 3. Forces produced in the braking analysis

BRAKING				
Charges	**Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg] Weight [Kg**	**Anchor Points**	**Force [N] Force [N] Force [N] Force [N] Force [N] Force [N] Force [N**	**Force at each point of Anchorage [N] Anchorage [N] Anchorage [N] Anchorage [N] Anchorage [N] Anchorage [N**
Pilot	80	6	1232.136	205.356
Engine	60	8	924.102	115.513
Suspension	30	16	462.051	28.878
Tank	18	4	277.231	69.308
Battery	15	4	231.026	57.756

When braking, the suspension exerts an opposing force to the movement of the vehicle producing reaction forces from both the weight of the rider and the mechanical components at their attachment points. Because of this, the suspension anchor points will be used to fix the structure in the simulation study. Figure. 7 represents the pre-process of the analysis.

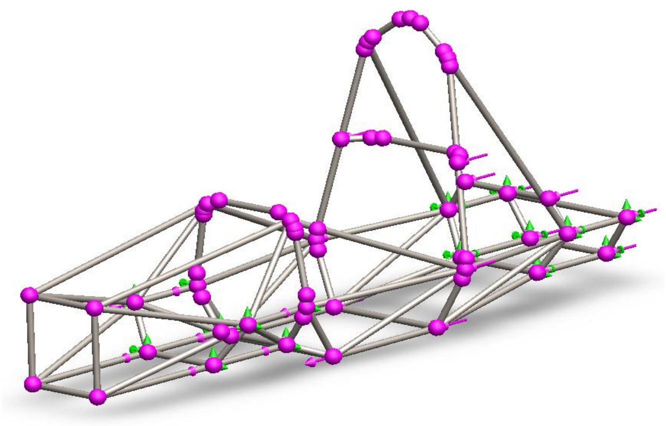

Fig. 7. Applied forces and attachment points for the braking analysis

Curve

The purpose of this analysis is to study the behaviour of the chassis when the single-seater takes a corner at high speed. At the instant when the vehicle turns, an acceleration is produced, which is composed of a tangential component and a normal component. The normal acceleration or also called centrifugal acceleration produces lateral loads on the structure at each of the attachment points of the mechanical elements so that the chassis deforms in the direction of the curve.

Equation 3 represents the calculation of the centrifugal force for a body of constant mass:

$$Fc = m * w^2 * r \tag{3}$$

Where:

FcIt is the centrifugal force.

m: It is the mass of the body.

wAngular velocity: It is the angular velocity.

r: It is the radius of curvature.

The angular velocity can be defined in terms of the speed and the radius of gyration (Equation 4).

$$w = \frac{v}{r} \tag{4}$$

Replacing Equation 4 in Equation 3 we have that:

$$Fc = m * \frac{v^2}{r} \tag{5}$$

In this way it is possible to know the centrifugal force in a more direct and simple way. For the calculation the total mass of the vehicle is taken as 255 kg, where this is the sum of all mechanical components, the weight of the chassis, the weight of the bodywork and the weight of the driver. A turning radius of 9 m will be taken, as this is the minimum radius indicated in the regulations and a speed of 60 km/h. Performing this calculation we obtain that:

$$Fc = 255\ kg * \frac{(16.67\ m/s)^2}{9\ m}$$

$$\boldsymbol{Fc = 7870.4\ N}$$

The acceleration of the vehicle is therefore $30.864\ m/s^2$which is equivalent to $3.15\ g$ Table 4 shows the lateral forces produced by this acceleration.

Table 4. Forces produced in curve analysis.

CURVE				
Charges	Mass [Kg]	Anchor Points	Force [N] Force [N] Force [N] Force [N] Force [N] Force [N] Force [N	Force at each anchor point [N].
Pilot	80	6	2472.12	412.02
Engine	60	8	1854.09	231.761
Suspension	30	16	927.045	57.940
Tank	18	4	556.227	139.057
Battery	15	4	463.5225	115.881

In the analysis, the chassis structure was restricted to the lower suspension anchor points, as the upper ones allow the steering rod to turn. The forces are applied in the direction of the X-axis and will be positive in the direction of the curve. Figure 8 illustrates the pre-process analysis.

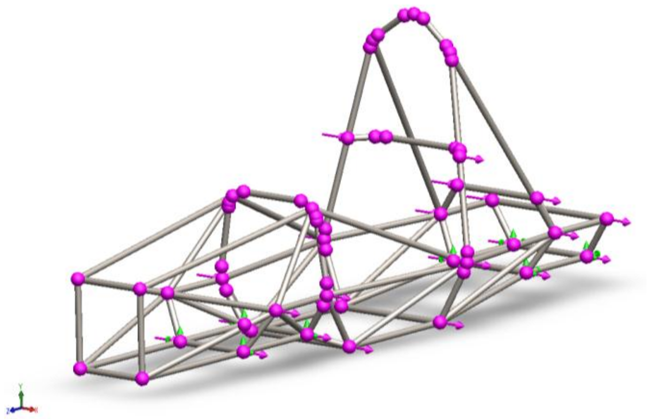

Fig. 8. Applied forces and clamping points for curve analysis

Torsional Analysis

The torsional analysis is the test that governs the behavior of the chassis, a light structure with good torsional rigidity significantly increases the performance of the vehicle at the time of the competition. In previous sections it was defined how the teams participating in the Formula SAE perform this test, for this the vehicle is fixed at the rear and then a torque is applied at the anchor points of the front suspension.

The force transmitted to the structure is that which is produced by applying a torque of $1000\ N * m$which is distributed to each of the front suspension anchor points deforming the chassis structure. Equation 11 allows the calculation of each of the forces applied to the chassis.

$$T = F * L \tag{6}$$

Where T is the torque, F is the applied force and L is the distance from the centre of the vehicle to each of the suspension anchor points. The respective calculations are shown in Table 5.

Table 5. Forces produced in torsional analysis at the front suspension anchor points

TORSION			
Node	Distance [m] Distance [m	Force [N] Force [N] Force [N] Force [N] Force [N] Force [N] Force [N	Force at each anchor point [N].
1	0.16	6250	781.25
2	0.16	6250	781.25
3	0.27472	3640.07	455.01
4	0.26496	3774.15	471.77
5	0.16	6250	781.25
6	0.16	6250	781.25
7	0.27472	3640.07	455.01
8	0.26496	3774.15	471.77

Once the forces were applied, the structure was fixed at the 8 nodes at the rear of the vehicle to perform the respective simulation. Figure 9 illustrates the pre-process of the analysis.

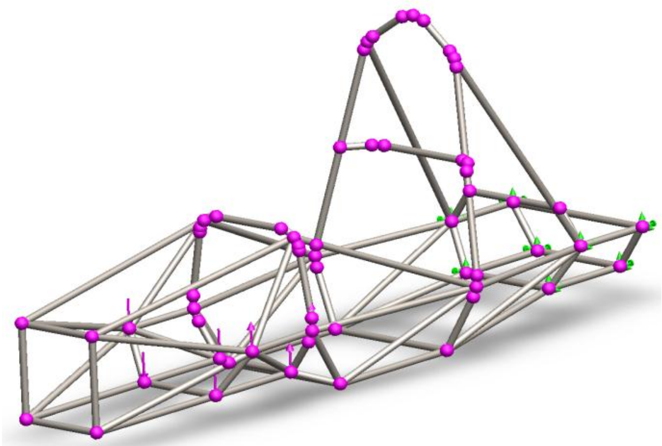

Fig. 9. Applied forces and clamping points for torsional analysis

2.2 Carbon Fiber Single-seater

The respective analyses for the single-seater chassis are carried out in a similar way to the tubular chassis. As it is a sandwich composite material, the core and the structure sheets must be parameterized specifying the material and thickness of both the core and the outer material. For the proposed model has been selected a honeycomb structure core of aluminum 5056 of 25.4

mm thick and 20 sheets of carbon fiber T300 3K of 0.25 mm thick for a total of 30 mm. As a last step, each carbon fiber sheet was assigned a location angle, the following pattern was used for the 10 elements on both sides of the core starting from the outside: 0°, 45°, 90°, 45°, 45°, 0. This is because in a real racing environment the loads present in the single seater are not unidirectional, so this configuration allows to better absorb the stresses present in the structure.

Frontal impact

The frontal impact test is carried out in accordance with the regulations. A force of 120 kN was applied in the direction normal to the frontal protection plane, distributed over the cross-sectional area of the frontal protection plane. The fixing points of the structure are the holes where the main arch is bolted to the single-seater. Finally, a mesh refinement was made with 1 mm elements in the area where the force is applied, in order to obtain the most realistic results in the simulation. Figure 10 illustrates the pre-process analysis.

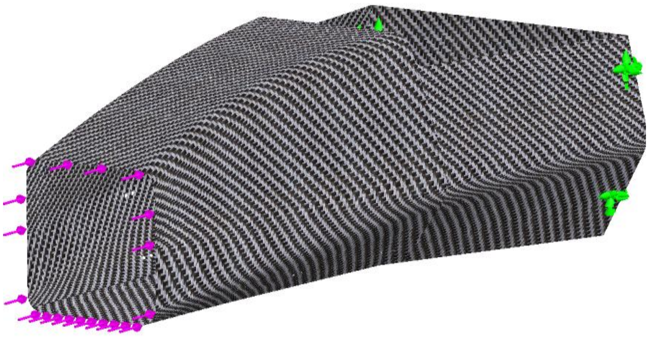

Fig. 10. Applied forces and attachment points for frontal impact analysis, single-seater

Side Impact

For this analysis the regulation indicates a force of 7 kN distributed along the side impact protection, which in this case is the side panel that extends from the front arch to the main protection arch. To fix the chassis, the holes where the chassis meets the main roll bar and the area where the engine is located are taken. Figure 11 illustrates the pre-process of the analysis.

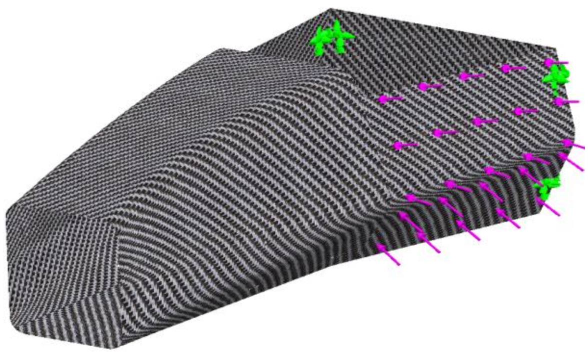

Fig. 11. Applied forces and restraint points for side impact analysis, single-seater

Front arc rollover

Years ago the regulations demanded that the front arch had to be bolted to the single-seater, so as not to put the integrity of the pilot at risk. But this made the single seater heavier and its rigidity decreased, so nowadays the vast majority of monohulls in the SAE formula do not use a front arch. To correct this, the thickness of the sandwich structure is increased in this area, which ensures the safety of the driver at all times and maintains a lightweight chassis with high torsional rigidity, either configuration is valid for the organization. The analysis was performed in its entirety as indicated by the regulations, the load was applied at the highest point of the front arc which was in the directions Fx = 6 kN, Fy = 5 kN and Fz = -9 kN, it should not generate a deformation greater than 25 mm avoiding the failure of the structure at any of its points. The attachment points of the chassis are the holes where it joins the single-seater with the rear structure and the main arch. Figure 12 illustrates the pre-process analysis.

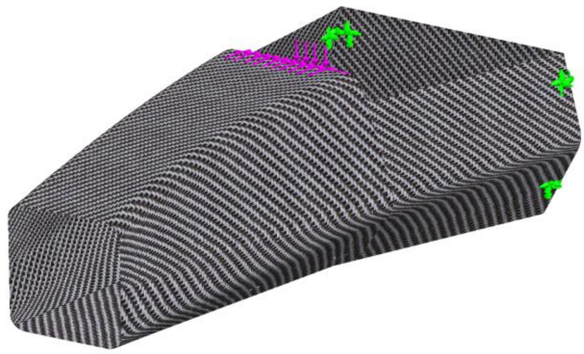

Fig. 12. Applied forces and attachment points for the front roll-over analysis, single hull

Acceleration

In the acceleration simulation study of the carbon fibre single-seater, the input calculations are performed in the same way as the tubular chassis. The total force will go in the opposite direction of vehicle motion and will be located on the outer walls of the single-seater. Since the axles of the suspension are responsible for transmitting the force produced by the acceleration, the structure will be fixed at the anchor points of this system and at the points where the single-seater joins the rear tubular structure avoiding displacement in the **X** and **Y** axes. Next, the pre-process of the analysis is illustrated (Figure 13).

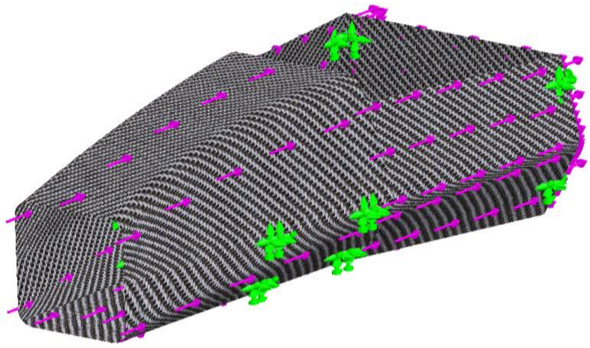

Fig. 13. Applied forces and attachment points for the acceleration analysis, single-seater

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 10 (2021), pp. 1005-1017

Braking

As previously indicated, due to the braking process the suspension exerts a counter force to the movement of the vehicle producing reaction forces both by the weight of the rider and each of the mechanical components throughout the vehicle structure. Because of this, in the simulation study the anchor points of the suspension and the holes where the single-seater is attached to the rear structure will be used to fix the chassis. Figure 14 represents the pre-process of the analysis.

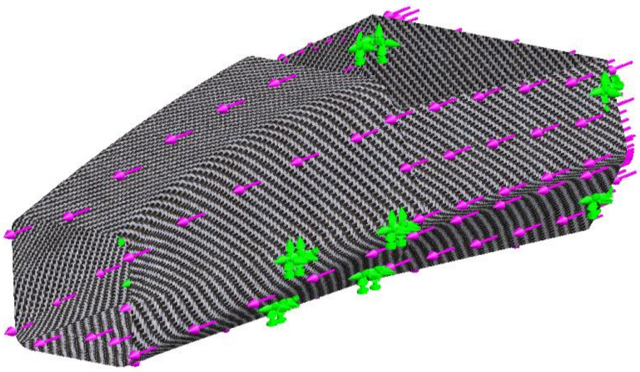

Fig. 14. Applied forces and attachment points for braking analysis, single hull

Curve

As with the tubular chassis, the purpose of the curve study is to study the behaviour of the chassis when the single-seater turns at high speed. In the analysis the chassis structure was restricted at the lower anchor points of the front suspension and at the points where the single-seater is assembled with the tubular structure at the rear. The forces are applied in the direction of the X-axis and will be positive in the direction of the curve. Figure 15 illustrates the pre-process analysis.

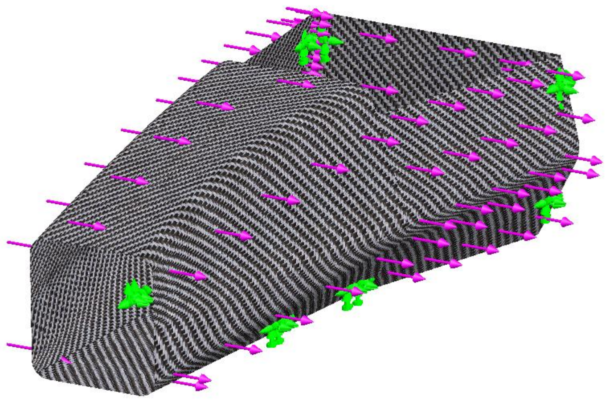

Fig. 15. Applied forces and attachment points for curve analysis, single hull

Flexural Analysis

The purpose of the bending analysis is to determine the behaviour of the single-seater under the stresses produced by each of the mechanical components of the vehicle and the weight of the driver in static condition. The analysis is performed in the same way as with the tubular chassis, simulating the loads produced by the heaviest components of the single-seater along its longitudinal axis, which are distributed in each of the anchor points of the chassis.

Torsion Analysis

To perform the torsion analysis on the carbon fiber single-seater, the loads were applied at the attachment points of the front suspension, which are the same as in the study of the tubular chassis. After this, the structure was fixed at the points where it is assembled with the main arch.

3. RESULT

Once the parameters of each analysis were established, the respective simulations were carried out, thus allowing to know the behavior of the structure. Each of the studies was carried out by applying the different materials proposed, comparing the results and determining the configuration with the best performance. As indicated in the regulations, the anti-overturn arches must always be made of carbon steel.

Front Impact

After the respective frontal impact simulation, the following results were obtained for each material in terms of maximum deformation and maximum stress.

Table 5 presents the results in a more detailed form for each simulation for the different chassis types.

Table 5. Frontal impact analysis results of the base prototype

FRONTAL IMPACT RESULTS				
Chassis	**Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa**	**Deformation** **Max [mm] Max.**	**Voltage** **Maximum [MPa].**	**F. S**
Steel 4130	460	1.138	108.1779	4.25
Aluminium 6161 T6	275	1.821	148.0849	1.85
Carbon Fiber	80	0.045	31.9	2.5

Each of the above configurations suffered a deformation of less than 2 mm, fully complying with the regulation, which specifies that the maximum allowable deformation must be less than 25 mm. The three structures presented a maximum stress lower than its elastic limit, so each configuration is valid, because this analysis by regulation means the worst possible case in competition for the vehicle.

Side Impact

Since the chassis structure is symmetrical the results obtained will be the same for the left and right side, so only the results of the left side of the chassis have been represented. The results for each structure in terms of deformation and stress were (Figure 16 to 21):

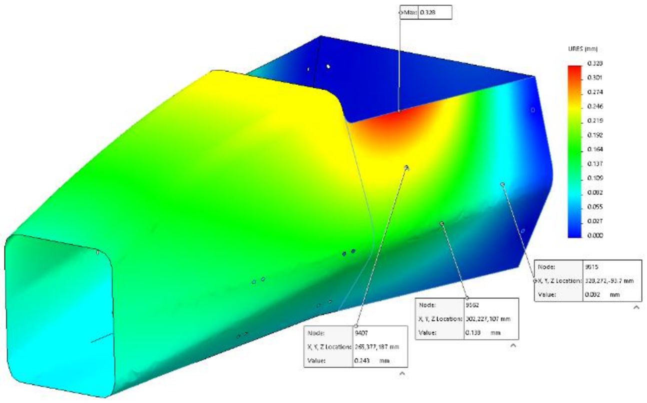

Fig. 18. Maximum Deformation in Lateral Impact Analysis, Carbon Fiber Single-seater.

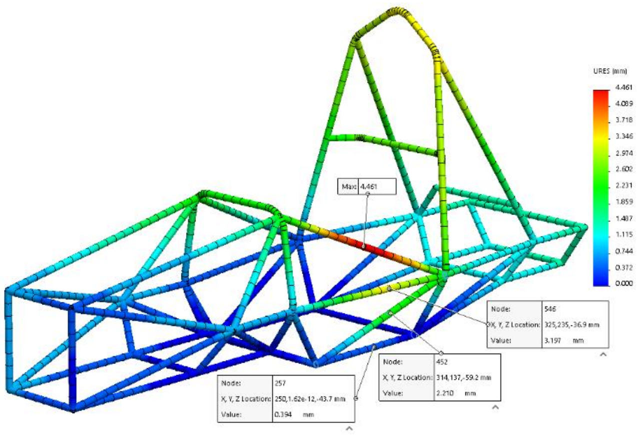

Fig. 16. Maximum deformation in the lateral impact analysis, 4130 steel tubular chassis.

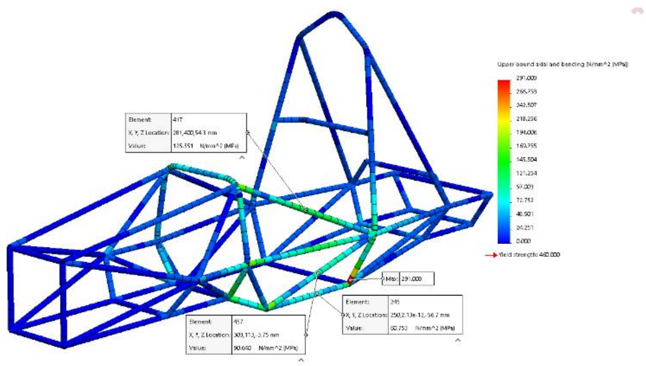

Fig. 19. Maximum stress in the lateral impact analysis, 4130 steel tubular chassis.

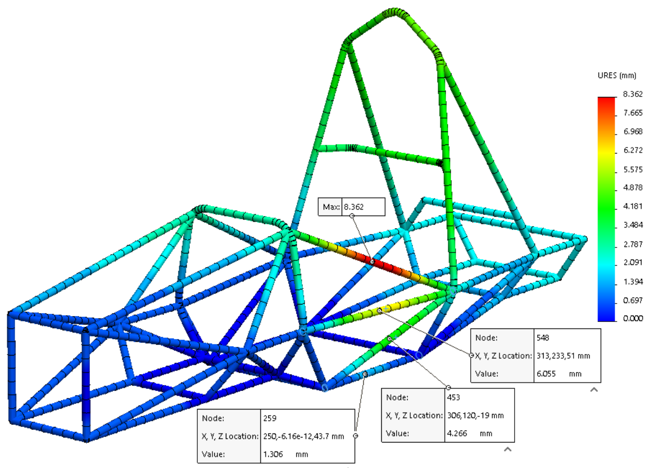

Fig. 17. Maximum deformation in the Lateral Impact analysis, 6061 T6 Aluminium tubular chassis.

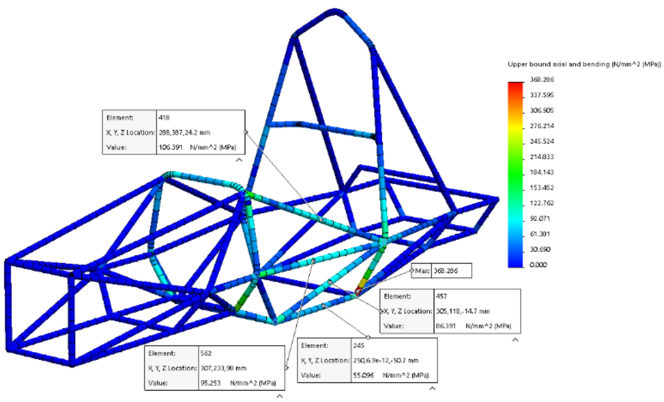

Fig. 20. Maximum stress in Lateral Impact analysis, 6061 T6 Aluminium tubular chassis.

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 10 (2021), pp. 1005-1017
© International Research Publication House. http://www.irphouse.com

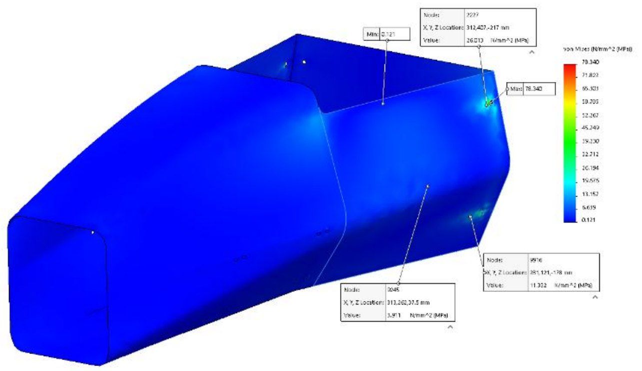

Fig. 21. Maximum stress in Lateral Impact analysis, Carbon Fiber Single-seater.

Table 6 contains the data obtained in the simulation.

Table 6. Side impact analysis results

SIDE IMPACT RESULTS				
Chassis	**Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa**	**Maximum Deformation [mm].**	**Voltage Maximum [MPa].**	**F. S**
Steel 4130	460	4.461	291.008	1.58
Aluminium 6161 T6	275	8.362	368.286	0.75
Carbon Fiber	*80*	*0.328*	*15.757*	*5.07*

However, the 6161 T6 aluminum chassis suffered a critical stress, failing in the lower area of the main arch where it joins with the rest of the structure, so it is not recommended to use it for the lateral impact protection of the chassis.

Overturning in main arch

Since the main arch is assembled by bolts to the single-seater, which should be only of steel structure, the study performed for the 4130 tubular steel structure is valid for the single-seater structure since the load is absorbed by the main arch and the maximum deformation and maximum stress will be present in this zone (Table 7). The results in the main arch analysis in terms of maximum deflection and maximum stress were:

Table 7. Overturning analysis results on main arch

Overturning - Main Arch				
Material	**Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa**	**Maximum Deformation [mm].**	**Maximum Tension [MPa].**	**F. S**
Steel 4130	460	34.1102	1014.8	0.453
Aluminium 6161 T6	275	54.9448	1390.8	0.197

The analysis of overturning in main arch yielded critical results. The deformations present were higher than the permissible value of 25 mm given by the regulations and the maximum stress greatly exceeded the tensile strength of the material. The main arch presented failure in the sections where it joins with the base of the structure and in the area where it joins with its supports. Taking into account the above, it is necessary to optimize both its geometry and the section of the tubular profiles used.

Front Arch Rollover

Table 8 shows the results of the frontal arch analysis in terms of deformation and maximum stress.

Table 8. Frontal arch overturning analysis results

ROLLOVER - FRONT ARC				
Material	**Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa**	**Maximum Deformation [mm].**	**Maximum Tension [MPa].**	**F. S**
Steel 4130	460	8.5377	729.6359	0.631
Aluminium 6161 T6	275	12.1106	738.626	0.372
Carbon Fiber	80	1.214	181.62	0.44

International Journal of Engineering Research and Technology. ISSN 0974-3154, Volume 14, Number 10 (2021), pp. 1005-1017

The results in the frontal arch analysis were very similar to the results of the main arch. Although the deformation remained within the established limit, the steel and aluminum structures exceeded the elastic limit of the material, failing at the attachment points of the supports of the main arch, so it is required to improve its geometry and section of the tubular profiles used. The carbon fiber single-seater presented good results in the front arch area, but failed in the points of union with the main arch, so it is of utmost importance to improve the design in this area, in order to guarantee the integrity of the pilot at all times.

Flexural Analysis

For the bending analysis, the following results were obtained in terms of deformation and stress, Table 9 shows each one in more detail.

Table 9. Flexural analysis results

FLEXURAL ANALYSIS				
Material	Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa	Maximum Deformation [mm].	Maximum Tension [MPa].	F. S
Steel 4130	460	0.0785	5.6591	81.28
Aluminium 6161 T6	275	0.2135	6.7977	40.45
Carbon Fiber	80	0.003	0.981	81.59

As can be seen in Table 29, the three structures presented an insignificant deformation which does not represent a risk for the chassis. In terms of maximum stress, each configuration suffered a stress of less than 10 MPa, so each structure is validated to withstand the loads produced by each mechanical component ensuring the integrity of the chassis and the pilot.

Acceleration

The results obtained in the acceleration analysis in terms of deformation and maximum stress for each material are represented in Figures 22 to 24 and Table 10.

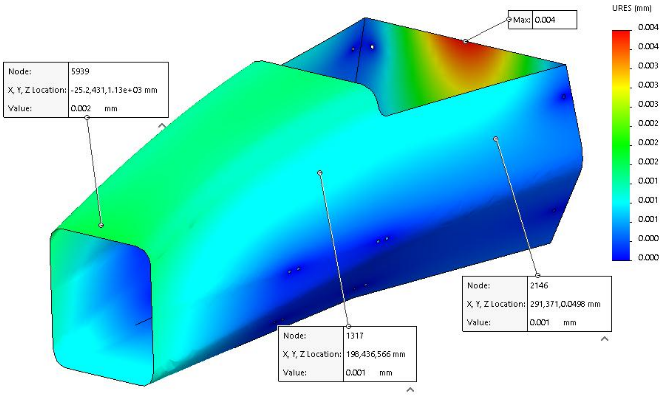

Fig. 22. Maximum Deformation in Acceleration Analysis, Carbon Fiber Single-seater

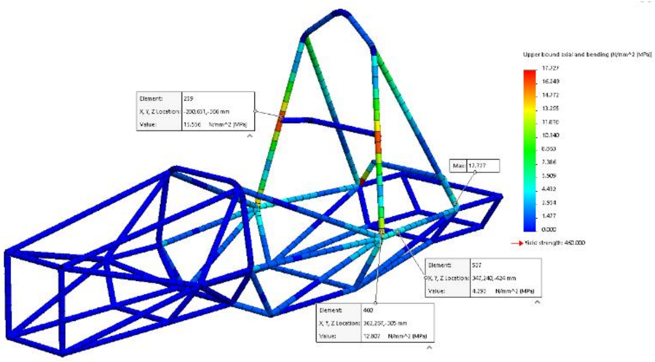

Fig. 23. Maximum stress in acceleration analysis, Tubular frame 4130 steel.

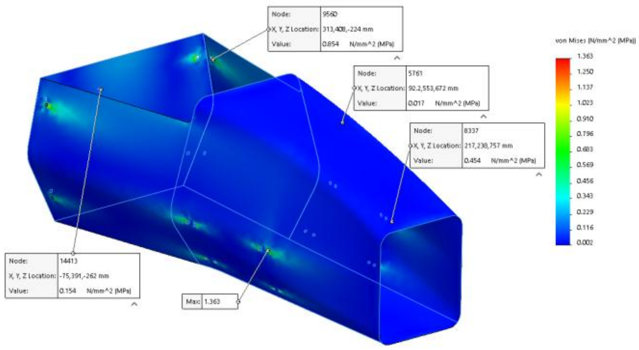

Fig. 24. Maximum stress in acceleration analysis, Carbon Fiber Single-seater.

Table 10. Acceleration analysis results

Acceleration				
Material	Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa	Maximum Deformation [mm].	Maximum Tension [MPa].	F. S
Steel 4130	460	0.234	17.727	25.94
Aluminium 6161 T6	275	0.312	17.741	15.5
Carbon Fiber	80	0.004	1.363	58.69

Due to the acceleration produced by the engine, each chassis presented a deformation of less than 1 millimeter in the points of union of the main arch and the anchor bar of the safety belt of the pilot's shoulders. The maximum stresses suffered are approximately less than 20 MPa, so each structure absorbs this type of load without problems.

Braking

The following results were obtained for the braking analysis, which are shown in Table 11.

Table 11. Braking analysis results

BRAKING				
Material	**Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa**	**Maximum Deformation [mm].**	**Maximum Tension [MPa].**	**F. S**
Steel 4130	460	0.413	31.27	14.71
Aluminium 6161 T6	275	0.55	31.297	8.79
Carbon Fiber	80	0.07	2.405	33.26

The maximum deformations produced in this analysis do not present any risk for the chassis, because the three structures did not present a deformation greater than 1 millimetre, so these values are negligible. In terms of maximum stress, each chassis suffered a stress well below its elastic limit, which are not a risk for the integrity of the tubular structure or for the pilot.

Curve

Since the structure is symmetrical, the results for both left and right hand curves will be the same, therefore, only the results produced by a left hand curve have been plotted, which are shown in Table 12.

Table 12. Results of Left Hand Curve Analysis

Curve				
Material	**Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa**	**Maximum Deformation [mm].**	**Maximum Tension [MPa].**	**F. S**
Steel 4130	460	4.1776	183.3896	2.5
Aluminium 6161 T6	275	7.9052	160.7361	1.71
Carbon Fiber	80	0.127	20.487	3.9

As shown in the above results, all three structures had adequate results in terms of the maximum stress produced, which does not represent a risk to the integrity of the structure. However, special care should be taken with the use of 6061 aluminum since, its maximum stress is close to its yield strength, so the probability of failure at a higher rate increases. The deformation as well as the maximum stress, is a factor of the speed with which the single-seater takes the curve, so this value should not be greater than 15% of the outer diameter of the tube used. In this case the maximum deformation is located in the upper part of the main arch and its supports, so a change in its geometry and thickness in the tubular structure will be considered. The carbon fiber single-seater presented a maximum deformation of less than 1 mm which is negligible, because of this the proposed geometry adequately supports the efforts produced when the vehicle rotates.

Torsion Analysis

The results for torsional analysis are represented in Table 13. In the case of the deformation, the deformation suffered by the chassis through the Y-axis is represented, since this value allows to know the angle of rotation of the structure.

Table 13. Torsional analysis results

TORSIONAL ANALYSIS					
Material	**Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa] Tensile strength [MPa**	**Maximum Tension [MPa].**	**Maximum Y-strain [mm].**	**Angle [°] Angle [°] Angle [°] Angle [°] Angle**	**Stiffness [N*m/°] Stiffness [N*m/°] Stiffness [N*m/°] Stiffness**
Steel 4130	460	128.284	2.945	0.6141	1628.3993
Aluminium 6161 T6	275	131.535	6.081	1.268	788.6435
Carbon Fiber	80	19.878	0.066	0.074	13513.51

As explained above, the participating teams manufacture the chassis structure looking for the best ratio of torsional stiffness and weight. Therefore, a vehicle with a stiffness greater $3000\ N*m/°$. The results of the analysis showed that the 6061 T6 aluminum structure has a too low stiffness, even when its weight is reduced by half compared to steel, so this material is discarded in the total use of the chassis structure.

Although the composite single-seater presented better results in terms of strength, stiffness and weight compared to the 4130 alloy steel tubular structure, the cost of raw material and manufacturing processes are very high compared to the latter. Therefore, it will be optimized as much as possible in terms of torsional stiffness and weight to ensure excellent track performance of the vehicle capable of competing with the carbon fiber single-seater.

4. CONCLUSION

By varying the orientation and thickness of the selected tubes, it is possible to obtain an infinite number of geometries with excellent performance for the tubular chassis, which facilitates the annual change of the structure just as indicated by the Formula SAE regulations.

At the end of the optimization process for the 4130 steel chassis, it is observed that the distribution of the stresses generated in each section of the structure becomes homogeneous. This is due to the correct dimensioning and position of the structural elements that support the dynamic loads of the single-seater.

The final results obtained from the proposed chassis through the finite element method proved that the designed structure is able to withstand all the loads present in a real racing environment. This thanks to the dynamic and static studies to which the structure was subjected. In each of the analyses a safety factor greater than 2 was presented, a torsional stiffness greater than 3500 N*m/°, a value that positively determines the behavior of the chassis in racing conditions, so the chassis reaches all expectations and is the benchmark for future work.

REFERENCES

[1] R. S. Borja Robalino y P. S. Morocho Rojas, «Análisis estructural mediante el método de elementos finitos del chasis del vehículo formula SAE eléctrico de la Universidad Politécnica Salesiana. Cuenca 2017», 2017, Accedido: oct. 29, 2021. [En línea]. Disponible en: http://dspace.ups.edu.ec/handle/123456789/15081

[2] «Formula SAE BRASIL - MATLAB & Simulink». https://la.mathworks.com/academia/student-competitions/formula-sae-brasil.html (accedido oct. 29, 2021).

[3] R. A. García Fernández, «Herramienta de simulación de la cinemática y dinámica del sistema de suspensión de un Formula Student mediante MATLAB - Simulink», 2016, Accedido: oct. 29, 2021. [En línea]. Disponible en: https://uvadoc.uva.es/handle/10324/18226

[4] S. S. Pérez, J. A. C. Magaña, C. R. López, y E. V. Negrete, «Modelado y simulación del automóvil en Matlab-Simulink-Carsim aplicado a la estabilidad del vehículo», *undefined*, 2018, Accedido: oct. 29, 2021. [En línea]. Disponible en: https://www.semanticscholar.org/paper/Modelado-y-simulaci%C3%B3n-del-autom%C3%B3vil-en-aplicado-a-P%C3%A9rez-Maga%C3%B1a/f6808661f16b065b01d79ac2d0d5e6484a69dd9c

[5] E. R. Mena Mena, «Optimización estructural topológica de las manguetas delanteras para un vehículo prototipo formula student mediante el análisis de elementos finitos», ago. 2017, Accedido: oct. 29, 2021. [En línea]. Disponible en: http://localhost:8080/xmlui/handle/123456789/2678

[6] «Análisis estructural chasis Formula SAE :: 4p-ingenieria», ene. 26, 2017. https://www.4p-ingenieria.com/l/analisis-estructural-chasis-formula-sae/ (accedido oct. 29, 2021).

[7] «Mecway - Finite Element Analysis». https://mecway.com/ (accedido oct. 29, 2021).

[8] J. Sáez Gómez, «Cálculo y diseño estructural de un sistema de suspensión de un Fórmula SAE», 2017, Accedido: oct. 29, 2021. [En línea]. Disponible en: https://idus.us.es/handle/11441/69383

[9] B. Trabalón Oller, «Análisis mediante el M.E.F. de un chasis de Formula Student», 2016, Accedido: oct. 29, 2021. [En línea]. Disponible en: https://repositorio.upct.es/handle/10317/5457

[10] F. Canut, L. Malcher, y A. Henriques, «Structural Analysis of a Formula SAE Chassis Under Rollover Loads», dic. 2015. doi: 10.20906/CPS/COB-2015-0837.

[11] M. L. Mohamad, M. T. A. Rahman, S. F. Khan, M. H. Basha, A. H. Adom, y M. S. M. Hashim, «Design and static structural analysis of a race car chassis for Formula Society of Automotive Engineers (FSAE) event», *J. Phys.: Conf. Ser.*, vol. 908, p. 012042, oct. 2017, doi: 10.1088/1742-6596/908/1/012042.

[12] R. Singh, «Structural performance analysis of formula sae car», *undefined*, 2010, Accedido: oct. 29, 2021. [En línea]. Disponible en: https://www.semanticscholar.org/paper/STRUCTURAL-PERFORMANCE-ANALYSIS-OF-FORMULA-SAE-CAR-Singh/df42360fa90e001331c9ebb5e804d5b384140bbb