XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

The Analysis of a SAE Formula Prototype Chassis Via Finite Element Method

Matheus C. Paulino[1], Koje D. V. Mishina[2], Monã G. G. V. Souza[3]

[1]Dept. of Mechanical Engineering, Federal University of Paraiba
maatheus.catao@gmail.com, 58051-900, Paraíba /João Pessoa
[2]Dept. of Mechanical Engineering, Federal University of Paraiba
kojemishina@gmail.com, 58051-900, Paraíba/João Pessoa, Brazil
[3]Dept. of Mechanical Engineering, Federal University of Paraiba
monangarcia96@gmail.com, 58051-900, Paraíba/João Pessoa, Brazil

Abstract. Engineering has always promoted efficiency linked with a safe and profitable design. In light of this, a tool has been growing, which is the CAE (Computer Aided Engineering). That tool models a design and tests it on a software via virtual prototypes. To put it simply, the CAE simplifies the development of a project, since having a computer-based study of the problem allows the developer to visualize the design without having neither cost with material nor with manufacturing the prototype. The engineer therefore saves time and money, besides not having accidental risk of a prototype's real failure, as it is all virtual. That is why the CAE perfectly fits in naval and aerospace engineering, because in those areas any minimal error can generate a fatality. Within the CAE technology, there are some subdivisions, the most famous are: CFD (Computational Fluid Dynamics), which studies transfer of heat and mass; and FEM (Finite Element Method), in which structural mechanics is studied. In the present work we will present the FEM in a chassis of a FORMULA SAE vehicle model, which will be modeled, made as discretization, simulated and by finalization an experimental table will be made, where the real results (experimental table) and the results obtained via software will be analyzed. Either the experimented and the simulated tables will demonstrate a very important factor in a vehicle chassis, that is the torsional stiffness. Thus, by analysing the torsional stiffness of the table, the engineer knows how a car will behave in a torsion, and such knowledge is crucial for the vehicle dynamics and pilot's safety.

Keywords: Computer Aided Engineering, Finite Element Method, Torcional Stiffness, SAE Formula.

1 Introduction

The SAE (Society Automotive Engineers) organizes and regulates student competitions at national and international level, among which there is the Formula SAE Brazil. This competition comprises the construction of a prototype type Formula, which involves from the initial conception, design, manufacture and assembly of the prototype. The entire project is developed by undergraduate students of engineering courses, under the coordination of a supervising professor. Students participating in this project have the opportunity to develop skills related to mechanical design, detailed budget, manufacturing, assembly and maintenance. Students can act in any sub-system, such as: suspension, brake, transmission, aerodynamics, engine, structure or the electrical part; always respecting the rules determined by the event organizer.

The frame (structure) is a component of fundamental importance in a vehicle, because in addition to containing all the other components of the prototype, it also accommodates the pilot. Thus, the chassis is essential for vehicle dynamics and for the safety of those who drive. Because of this, the chassis is the component that has the largest number of rules and requirements in its design and dimensioning. In this context, torsional stiffness is one of the main parameters when designing the chassis.

To analyze the chassis torsional stiffness, initially used the Finite Element Method during the prototype design phase, in order to obtain a theoretical Torsional Stiffness. And afterwards when the chassis was manufactured, an experimental bench was built to obtain the experimental torsional rigidity of the prototype.

2 Theorical Development

2.1 Torsional Stiffness

The stiffness constant for a vehicle structure is something that is taken into account during a design phase. The relationship between weight and rigidity is always sought, so much that an optimal car is light and rigid, but in practice this relationship is inversely proportional, since for the same material the more mass it has, the more rigid it will be.

A vehicle prototype structure has diverse types of stiffness, such as: for vertical and lateral flexion, torsion and axial deformations. But, according to Costin and Phipps (1974) when a chassis has a good performance in a torsion, it has a broad performance related to flexion and other deformations [1]. In addition, a structure with a constant low for torsional deformation, suffers from dynamic effects and vibrations; greater damage and probability of failure due to fatigue; in addition to poor handling, due to possible lack of control caused by greater deformations and displacements at the suspension points. On the other hand, a design with high torsional deformation constant causes excess weight and a concentration of stresses that can cause fracture s and permanent deformations. In view of this, SAE created a table that relates the type of vehicle with torsional stiffness (Table 1) [2].

Table 1. Chassis Torsional Stiffness for different groups of vehicles (SAE)

Vehicle	Chassis Torsional Stiffness (Nm/°)
Formula SAE	1000-5000
Passenger car	5000-20000
Winston Cup Racing Car	15000-30000
Sports car	15000-40000
Formula One car	10000-10000

2.2 Torsional Stiffness via Finite Element Method

During the design phase, a way to optimize the process is by using Finite Element Method (FEM) simulation software, which is a numerical method that discretizes the structure into small elements, which are divided by us. Thus, was calculated the rigidity of each element, in order to have a global analysis, in the case of the structure as a whole. Within the scope of FEM, there are the three main types of elements: the beam element (1D), the plate element (2D) and the solid element (3D) [3].

For this application, the simplest element, the Beam Element, will be used, as it has the main degrees of freedom, for torsion, flexion and axial deformations, besides this element has low demand for computational resources, and even with its limitations, fits perfectly as needed, because for a calculation of a stiffness are important the loads and deformations, besides that the element 1D is more used when it is necessary to know how to deformations of a structure.

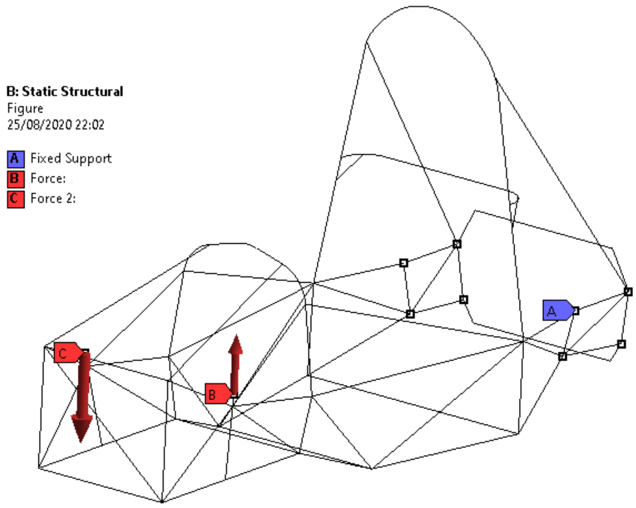
Within this context, the student version ANSYS WORKBENCH simulation software was used to determine torsional stiffness, with the following boundary conditions: binary load at the front suspension points and fixation at the rear suspension points (Figure 1). As the Torsional Stiffness is given in Newton meter per degree (Nm / °), the values of the torsor moments (Equation 1) and Angles (Equation 2) were removed, which was varied as torque loads and after that the values were plotted, the moments on the ordinate axis and the abscissa angles. Then, was used the method of the line equation, to obtain the value of the constant in Newton meter per degree.

$$Kt_{simulation} = 2 \times P \times L \quad (1)$$

$$\theta_{simulation} = atg\left(\frac{\Delta A + \Delta B}{2 \times L}\right) \quad (2)$$

P: Load for binary;
L: Distance from the suspension points;
ΔA and ΔB: Maximum and minimum deformation respectively.

Figure 1. Boundary Conditions of Simulation



2.3 Torsional Stiffness via Experimental Bench

After the Manufacture of the Prototype Chassis, an experimental bench (Figure 2) was made to verify the torsional rigidity of the real prototype. Beams, a pivot, weight plates and dial indicators were used so that the results obtained were befitting with those obtained via software.

One beam was engasted in order to fix the rear suspension point, and the other was pivoted so that when it moves, due to the weight of the plates, it generates a twist in the chassis, the dial indicators were used to obtain the displacement of the beam to calculate the angles then calculating the stiffness.

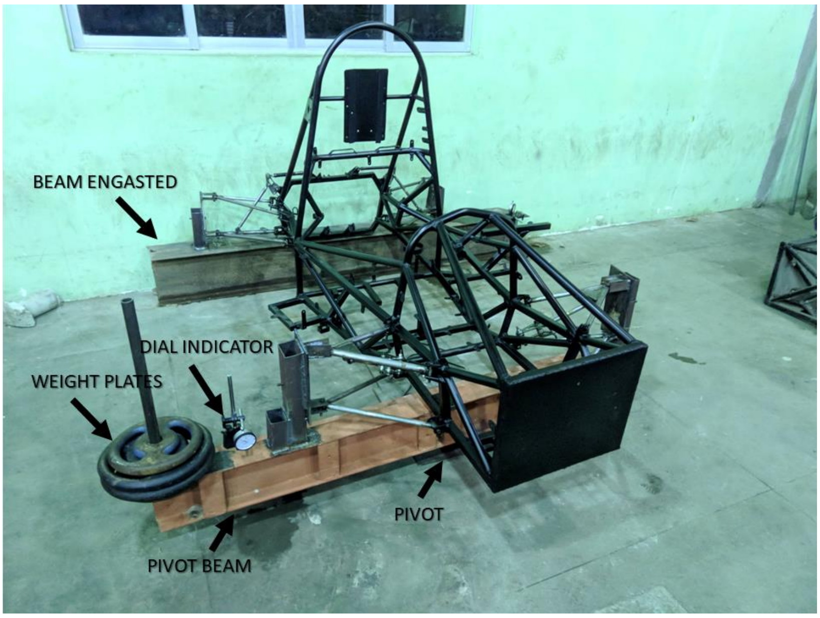
Thus, as the simulation, the stiffness is obtained in a similar way, the values of the torsion moments (Equation 3) and angles (Equation 4), varying the mass, and finally a graph is generated, which by the linear equation method is defined as Torsional Stiffness.

$$Kt_{real} = P \times L_1 \quad (3)$$

$$\theta_{real} = atg\left(\frac{\Delta y}{L_2}\right) \quad (4)$$

P: Load for binary;
L_1: Distance from the pivot to the load (weight plates);
L_2: Distance from the pivot to the dial indicator;
Δy: Beam displacement measured by the dial indicator.

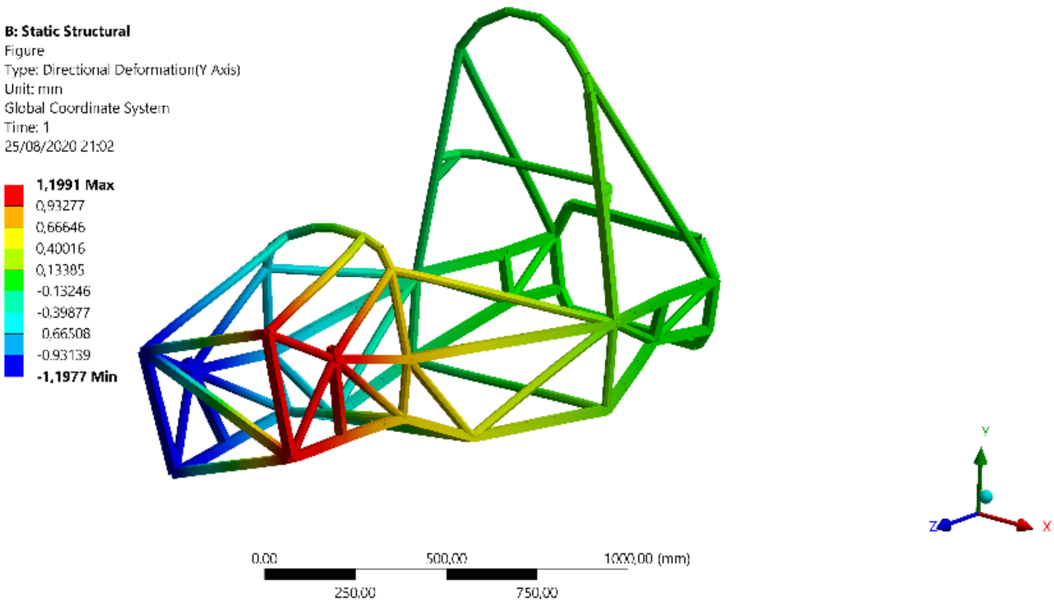
Figure 2: Experimental Bench



3 Analysis and Results

As shown in Table 1, Formula SAE vehicles must have torsional stiffness between 1000 to 5000 Nm/°, and for this purpose, still in the design phase, analyzes were made using the finite element method, so that the chassis has such a requirement. For that, the ANSYS Workbench software was used, with the boundary conditions shown in Figure 2, which varied the torque load from 500 to 1000N, adding 100 N to each simulation and as a result the directional deformations on the Y axis were taken, Delta A being the maximum and Delta B the minimum, for then used the Equation 2. As in the simulation shown (Figure 3).

Figure 3: Simulation for a Binary of 800N

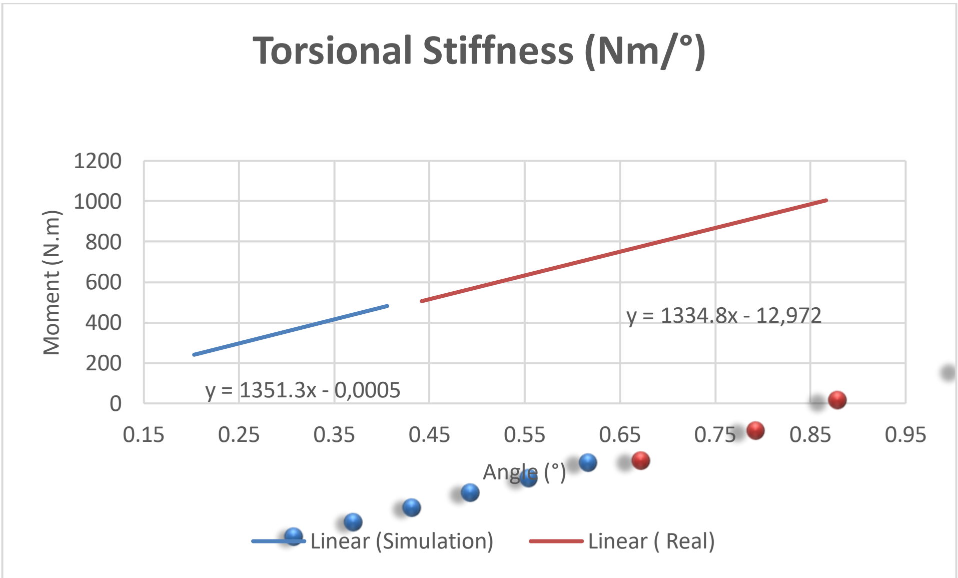


Thus, an Angle x Moment graph was drawn up using Equations 1 and 2, which obtained the torsional stiffness value of the simulated model using the linear equation method.

Subsequently, after the prototype was elaborated in a similar way, it was made with the test bench, using Equations 3 and 4, another graph was generated from the data obtained experimentally, which varied the mass from 50 to 100kg, adding 10kg to each test, with a 1 meter lever arm and the displacement data measured by the dial indicator. Finally, the linear equation method was used to obtain the real torsional rigidity of the prototype.

Then, a graph was plotted with the two lines and obtained their respective equations (Figure 4).

Figure 4: Simulated and real torsional stiffness



4 Conclusion

As shown in the graph shown at Figure 4, it appears that during the design phase the analysis by the FEM obtained a model stiffness value of 1351.3 Nm / °, while in the experimental analysis on the chassis a stiffness of 1334.8 was obtained. This corresponds to a relative error of 1.22%, which is considered acceptable, and may be linked to the type of welding and the chassis gauging, since prototyping was performed manually. Another relevant aspect is that, in both analyzes, the structure meets the criteria established by SAE, as it is in the range of 1000 to 5000 N.m /°, which shows that the chassis design was successfully carried out.

References

[1] Costin, M. and Phipps, D., “Racing and Sports Car Chassis Design, Batsford”, 1974, ISBN 978-0713404586.
[2] E. Sampo, A. Sorniotti and A. Crocombe, "Chassis Torsional Stiffness: Analysis of the Influence on Vehicle Dynamics". In: University of Liverpool. Publisher, 2010.
[3] A. A. Filho "Elementos Finitos - A base da tecnologia CAE”. In: Editora Érica (6th ed). Publisher, 2018.