COBEM-2017- 0856
MODELING AND DYNAMIC ANALYSIS OF A SINGLE-SEAT OFF-ROAD VEHICLE'S TUBULAR CHASSIS

Eduardo Ataide de Oliveira[1]
Instituto Federal de Pernambuco – IFPE, Estrada do Alto do Moura, Distrito Industrial III, CEP:55040-120, Caruaru – PE
eduardo_ataide@hotmail.com[1]

Abstract. This work was accomplished through the study using finite element method, which consists of dividing a body creating a mesh consisting of several parts or elements formed by us. Technique allows complex calculations of deformation structures, displacement and stresses in the structure, values or approximate solutions of real, independent of your shipment. The chassis is the basic component of an automotive vehicle. It consists of an internal structure that receives all the other components of the vehicle and must withstand all mechanical loads, to ensure safety of occupants, the correct closing of doors and the operation of the Steering and suspension systems. (MILLIKEN, 1995). The model to be studied was developed before a security regulation set by the SAE (Society of Automotive Engineers), with the help of a software of CAD (computer aided design), thus, it was possible to make the chassis modeling applied to the vehicle type baja. With the model ready, began the study of simulations performed is a software for CAE (Computer Aided Engineering), in which it has been assigned to the type of material, loads (forces), type of mesh, and fixing conditions of the structure, supposedly where they will be assembled front and rear suspension. To assign the loads on the structure analysis was used the Australian ADR 59 Protocol.

Keywords: Finite Elements; Deformation; Chassis; Loading; Baja

1. INTRODUCTION

Considering that every structure has a bending stiffness, torsion or axial. The structure of an automotive vehicle is no different in relation to your study as a whole, however, studies such as flexion and torsion stiffness of a vehicular chassis is extremely important on the dynamic behaviors to the same is submitted.

The structure modeling was performed in a software of CAD (Computer-Aided Design) and this milestone is just as important as held in the CAE software (Computer-Aided Engineering), where it will be done the dynamic study of the chassis. To obtain the results closer to reality, the chassis modeling presents all the characteristics of the actual structure, such as: the type of material and its properties.

To perform dynamic analysis of chassis was used the finite element method to obtain safer and reliable results, through CAE software. The finite element method is a numerical approximation where partial differential equations can be solved to approximate mode (Fish; Belytschko, 2009, p. 1).

"The idea of prime importance for the understanding of the finite element method, is to divide the body into various elements, connected through us. The set of nodes and elements in a play or a body is named. The greater the number of elements and we, more refined is the mesh and the better the result obtained (Fish; Belytschko, 2009, p. 1).

The finite element method has as its starting point the relationship between nodal forces and nodal offsets for each individual element. This idea is fundamental and is related to the concept of stiffness (Alves Filho, 2000, p. 19).

In this way, we can say that the rigidity of the whole structure depends on the stiffness of each of its elements. Thus, it is possible to mount the stiffness matrix of the structure from the rigidity of each array element. (Alves Filho, 2000, p. 21).

The objective of this study sought to improve the chassis (2016 prototype) of the Bajagreste Team, that it participated in the baja SAE competition step Northeast and national championship. In this way it was possible to develop a lighter structure, resistant and safe for the pilot.

2. METHODOLOGY

A few steps to the realization of the simulation of structural analysis should be followed. Being able to obtain more accurate results and satisfactory. These procedures are represented in the following flowchart:
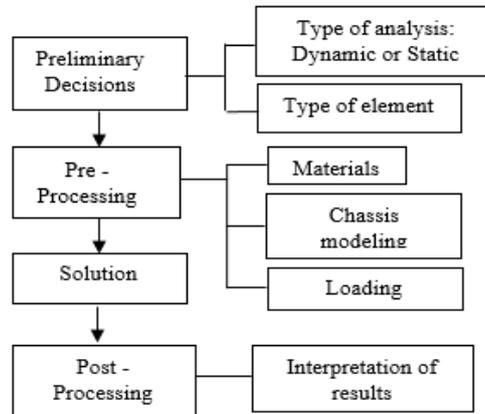
Flowchart for analysis by finite element method

Following this diagram it is possible to define what will be accomplished at each stage:
- **Preliminary Decisions –** This step is where we must define what kind of analysis will be performed as: dynamic or static. It is also where we define the model that we're going to study, and the type of element that will be used for discretization of the model;
- **Pre-Processing-** In the preprocessing is set the material, creating the geometry, mesh generation, and the application of shipments;
- **Solution –** Is the solve, the analysis is when the CAE software starts to run the SIM;
- **Post Processing-** The post-processing step is one of the most important, is where the skill of the analyst is crucial to interpretation, verification and validation of the results.

2.1 Preliminary Decisions

All work was done in front of a dynamic analysis, applied through a regulated Protocol, the following will be presented the concepts used in this work, conceptualizing what is a dynamic analysis and your behavior as a function of a structure (chassis). Will also be presented the types of elements, and the element that was used in the analyses.

2.1.1 Dynamic Analysis

In dynamical systems, the response of the structure varies over time, added to the response of the structure the effects of inertia forces and damping. Dynamic loads, unlike static, in which shipments are the magnitude, direction and position may vary over time (Furtado, 2013).

According to Alves filho (2005), the equations that describe the dynamic behavior of the many nodes of the structure is expressed by the matrix equation:

$$[M] \times \left\{ \ddot{U} \right\} + [C] \times \left(\dot{U} \right) + [K] \times \{U\} = \{F(t)\} \tag{1}$$

In which [M], [C], [K] are the n x n matrices of stiffness and damping mass, respectively. F (t) is a vector of external load acting on the structure of dimension n and Ü, U and U (respectively with the equation 1) represent the vectors of accelerations, velocities and nodal displacements of the structure, also n dimension.

In a dynamic analysis, as it is a static analysis to obtain the results of deformation of a structure is necessary to know the behavior of the meshes. Knowing that a mesh exists multiple nodes and elements, in which these with the application of a force suffers reactions, causing displacements and deformations in the nodal element. The big difference in dynamic study of a structure that variables such as acceleration and speed are of the utmost importance.

Knowing that the entire structure has a degree of freedom, similarly the more degrees of freedom have a structure therefore the higher your amount of vibration modes. A structure for nature has natural frequencies and vibration modes, this is called modal Analysis.

According to Alves Filho (2005), modal analysis reflects the basic structure and dynamic behavior is an indication of how it will respond to the dynamic loading agent on it.

Second Furtado (2013), modal analysis is performed considering the structure subjected to free-vibration damped and aims to find the natural frequencies and modes of vibration of a structure, as it represents the equation 2.

$$M \times U + K \times \ddot{U} = 0 \tag{2}$$

When a structure is away from your position of equilibrium with the deformed configuration of one of your natural modes of vibrations, she vibrates left to herself with that configuration mode and with a frequency that vibrate mode feature. (Alves Filho, 2005, p. 158).

Any study by the finite element method, is accomplished through the rigidity of the element, and each element has x number of degrees of freedom. In a system with n degrees of freedom can be represented by the stiffness and mass associated with each degree of freedom.

According to Alves Filho (2005), each degree of freedom features a vibration free movement similar to body-spring system, vibration free process harmonically, i.e. each degree of freedom during a free vibration structure performs a harmonic motion simple.

So we have a matrix form compresses all the offsets associated with n degrees of freedom, but also the amplitudes associated with n degrees of freedom movements, writing as follows:

$$\{U(t)\} = \{U_0\} \times \sin \omega \tag{3}$$

Being {U (t)} the column array that contains all components of displacements associated with n degrees of freedom of the structure in a time t, ω being the natural frequency and {U0} is the matrix column that contains all maximum offsets associated with n degrees of freedom, i.e. the amplitude of oscillations associated with varying degrees of freedom. The equation of dynamic equilibrium, which is given as follows:

$$([K] - \omega^2 [M]) \times \{U_0\} = 0 \tag{4}$$

According to Alves Filho (2005) is equation represents the path to solve the first step of the calculation of dynamic response, i.e. the calculation of natural frequency of the structure.

Where the natural frequency of vibration of a machine or structure match the frequency of the external excitation, a phenomenon known as resonance, resulting in excessive displacements, being liable the collapse of the system. Due to the devastating effect that vibrations can cause machinery and structures, vibration testing become a standard procedure in the design and development of most engineering systems. (Rao .2008, p. 8).

Therefore, it is crucial in mechanical and structural designs submitted to time-varying loads, which is undesirable behaviour of the structure when requested. One way to avoid the structure work on your natural frequency is changing the mass or the rigidity of the system, through changes in geometry, for example, to offset the natural frequencies to higher values, away from the frequency bands in which the equipment works. (Moura, 2010).

2.1.2 Element Type

According to Norton (2010), the elements can be one, two or three dimensions, or elements of line, volume area, respectively. They can also be of different "orders", a term that refers to the order of the function used to interpolate the offset inside the element.

Figure 1 shows some types of the most common elements used in computational analyses by the finite element method.
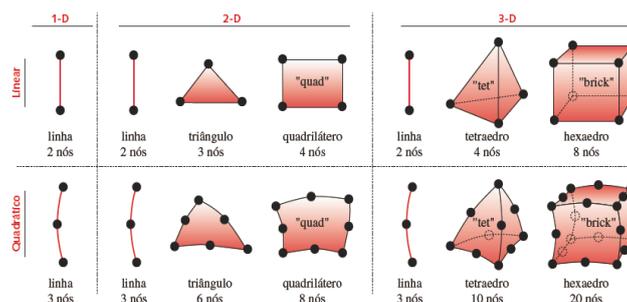

Figure 1. Finite element types
Source: Norton (2010)

Plans elements has a smaller amount of degree of freedom in relation to three-dimensional solid elements. For example a linear triangular or tetrahedral element, the TIMESHIFT feature via element is a line or a plane and your constant deformation. This makes them excessively strict.

Knowing that the tension is calculated from deformation and properties of materials, then the triangular and tetrahedral linear elements have values of constant tension. Elements such as the hexahedron and tetrahedra have offset parabolic functions and linear deformation and stress distribution in the element, which allows better estimates stop tensions (Norton, 2010, p. 487).

According to Norton (2010), experts do not recommend the use of triangular elements and three or four tetrahedra us because of inaccurate estimates for the tension and stiffness. Soon, best estimates are achieved using a four-Quad or an eight-node hexahedron, which have linear deformation on element.

However, due to computational limitations was chosen the tetrahedron element element, for resolution of the mesh. Knowing no better element type for this type of study, sought the best solution to obtain a more accurate result, this way was applied further refining the mesh, decreasing the space between each element.

2.2 Pre – Processing

2.2.1 Materials

The material used in the construction of the chassis must follow the regulation of the leaves. The same reports that steel must have a minimum carbon content of 0.18%. The prototype to be studied will be modeled in SAE 1020 Steel used in the manufacture of the structure.

The SAE 1020 Steel is one of the most common carbon steel in the industrial branch, being fairly used as steel for carburizing, with excellent cost benefit compared with most steels connected for the same purpose. The same features as good weldability, low mechanical strength.

Table 1. Mechanical property informed by manufacturer catalog.

Mechanical Properties	SAE 1020 steel Laminate
Limit of tensile strength MPa (N/mm ²)	450
Limito f yield MPa (N/mm²)	330
Module of elasticity (GPa)	205

Source: Gerdau Steels 2003 Edition manual

In addition to the 1020 steel it is important to know the mechanical characteristics of the tube, such as stiffness and resistance to flexion flexion. To calculate the bending rigidity do the product between the module of elasticity of the material (and) and moment of inertia (I). Regulation 7 RBSB offered by SAE States that the minimum value of the bending stiffness for strong tube is 2.79 x 109 [N.mm2] and additional tubes 1.0563 x 109 [N.mm2]. We also calculate the resistance to flexion which is defined as:

$$\frac{S_y \times I}{c} \tag{5}$$

where "Sy" is defined to be the limit of material flow, "I" moment of inertia and "c" (RADIUS) the distance from the neutral line until the normal of the tube. Soon, to the strong tube bending resistance must be greater than or equal to x 103 396.8 [mm] and additional tubes x 10 150.1233 [mm]. Thus we can calculate the bending rigidity and resistance to bending of tubes

- Strong tube (31,75 x 1,9 mm) – E x I = $(205 \times 10^3\,[\,N/mm^2])$ x $(19925,17417[mm^4]) = 4,084 \times 10^9[\,N/mm^2]$
$$\frac{S_y \times I}{c} = 464,397 \text{ [N.mm]}$$

- Additional tube (25,4 x 1,2 mm) – E x I = $(205 \times 10^3\,[\,N/mm^2])$ x $(6695,043706\,[mm^4]) = 1,3724[\,N/mm^2]$
$$\frac{S_y \times I}{c} = 195,052 \text{ [N.mm]}$$

- Additional tube (25,4 x 0,9 mm) – E x I = $(205 \times 10^3\,[\,N/mm^2])$ x $(5204,58301\,[mm^4]) = 1,0667\,[\,N/mm^2]$
$$\frac{S_y \times I}{c} = 151,629 \text{ [N.mm]}$$

2.2.2 Chassis Modeling

The modeling of the structure is a crucial step to achieve a good result of dynamic or static analysis of structure. The model to be studied must be designed as real as possible what you want to be manufactured. In the modeling should take into consideration the type of material that the chassis will be built, in the case of the structure being studied is the tubular type, thus it is necessary to use the correct dimensions of the tubes that make up the chassis. In addition to the type of material and its dimensions, a rather critical point are welded joins, those joins because the solder will present stress concentrators, change the microstructure of the material, and a fragile region.

It was developed two models of chassis model 2015 and 2016 model to be studied and compared the models, the structures were made in a CAD software (SolidWorks) they are represented below:

The 2015 model was built with SAE 1020 Steel tubing, the tube has 31.75 mm outside diameter and the thickness of your wall is 2.00 mm. Chassis is being represented in Figure 2.
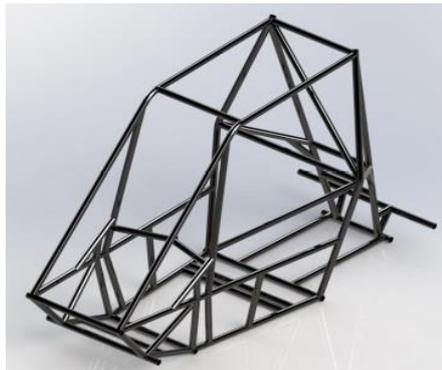
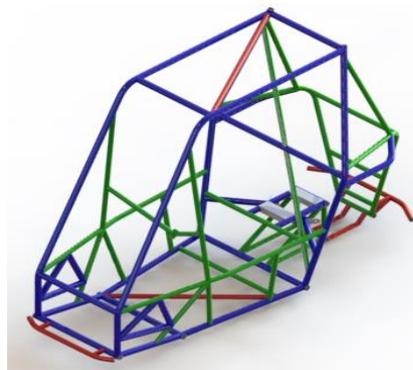Figure 2. Tubular chassis model 2015
Source: Author

Figure 3. Tubular chassis model 2016
Source: Author

The 2016 model as can be seen in Figure 3 above, was built with SAE 1020 steel tube. For better optimization of the design was used with different dimensions of the 2015 model. We use tubes with the following dimensions (outer diameter and wall thickness respectively): Blue-31.75 x 1.9 mm, green-25.4 x 1.2 mm and "Reds"-25.4 x 0.9 mm. larger diameter tubes are considered strong tube, and for smaller diameter adicionai tubes s.

Due to results of previous simulations, some behaviors were observed in the structure, as deformations resulting from the natural frequency of the structure when it was performed a dynamic study of modal analysis, trying to minimize these deformations was placed on the model 2016 2 crashes on a diagonal on the floor and another on the top of the cage. These locks allows the reduction of the natural frequency of the cage, therefore leaving more rigid and avoiding undesirable failures as a result of resonance or fatigue fracture of cage, due to cyclic efforts in which it is submitted. In this way we can analyze is reduced using the following equation.

$$f = \frac{1}{2\pi}\sqrt{\frac{K}{m}}$$

(6)

As we can see the equation 6, the frequency can be determined by mass or stiffness. As, the increase of the mass is not interesting for the chassis design, consequently we increased the rigidity of the chassis. Observing, the equation (6) with the increased stiffness, consequently increasing the frequency, but anyway this increased frequency does not contribute to any kind of failure in structure, being highlighted in the results.

2.2.3 Mesh

The mesh was generated in own software CAE-Ansys, Figure 4 (a), presents the parameters used in the generation of the model to be studied, we can highlight the relevance of which was set at 50, the range is (-100 to 100), the Centre of relevance set to medium as well as the center of the expansion angle (angle of opening between elements). Figure 4 (b) displays the number of nodes and elements, generated in the geometry to be studied.

Defaults	
Physics Preference	Mechanical
Relevance	50
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Medium
Minimum Edge Length	0,396520 mm
Inflation	
Use Automatic Inflation	None
Inflation Option	Smooth Transition
Transition Ratio	0,272
Maximum Layers	5
Growth Rate	1,2
Inflation Algorithm	Pre
View Advanced Options	No
Patch Conforming Options	
Triangle Surface Mesher	Program Controlled
Patch Independent Options	
Topology Checking	Yes

Advanced	
Number of CPUs for Parallel Part Meshing	Program Controlled
Shape Checking	Standard Mechanical
Element Midside Nodes	Program Controlled
Straight Sided Elements	No
Number of Retries	Default (4)
Extra Retries For Assembly	Yes
Rigid Body Behavior	Dimensionally Reduced
Mesh Morphing	Disabled
Defeaturing	
Pinch Tolerance	Please Define
Generate Pinch on Refresh	No
Automatic Mesh Based Defeaturing	On
Defeaturing Tolerance	Default
Statistics	
Nodes	513775
Elements	266219
Mesh Metric	None

Figure 4 (a). parameters for the mesh resolution.
Source: Author

Figure 4 (b). parameters for the mesh resolution.
Source: Author

Figure 4 shows the mesh generated in the structure. For best viewing was extended a part of the model as shown in Figure 5, where we can observe the element type that was generated in the cage.
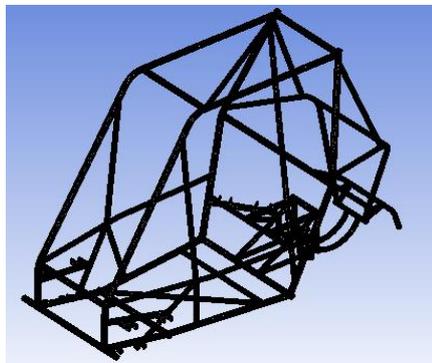
Figure 5. generated Mesh in structure.
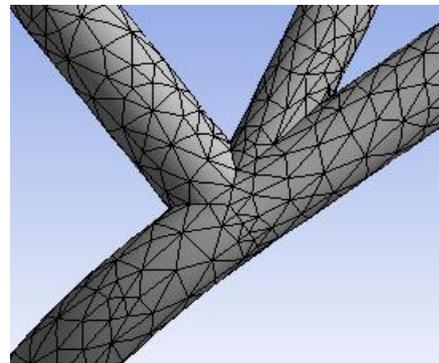**Source:** Author

Figure 6. expansion of the welded joint.
Source: Author

2.2.4 Application of Loads

The application of loads defined in this work will be performed using the Australian ADR 59 Protocol. The same lays down specific conditions for shipments in rollover conditions and tipping. The analyses of tipping and overturning are required due to the adverse conditions that Baja is exposed, so we can guarantee the quality of the built structure and more to ensure the safety of the pilot.
The Australian ADR 59 Protocol, defines equations for determining loads on vertical components, front and side. These equations are defined in the following way as shown in Table 2.

Table 2-equations for the vertical loads, horizontal and lateral.

Type of loading	Type of loading	Type of loading	Type of loading
Equation	4.g.M	1, 5.g.M	1.g.M

Source: Lottermann, (2014).

Taking as parameter of car 2015, where the same had a mass of about 240 Kg, we must also consider the weight of the pilot, admitting that an adult has approximately 70 kg pilot. The total mass of the vehicle is of 310 kg. Thus, we can determine the loads exerted in the cage. As is shown in Table 3.

Table 3 - applied in the structure, found using the Australian ADR-59 protocol

Type of loading	Type of loading	Type of loading	Type of loading
Equation	12,1644 KN	4,56165 KN	3,0411KN

* Note: considering the seriousness as: 9, 81m/s^2

2.3   Boundary Conditions

According to the Australian Protocol for the pilot to remain safely within the structure, the cage can present a maximum deflection in the three dimensions (vertical/front/side) up to 100 mm (Leku .2014). The applied loads have been defined in table 3.

The bezel of the structure will be made in the front and rear suspension. The balls represent the masses suspended as the pilot, engine, transmission. Being represented in Figure 7 (a) and (b) respectively.
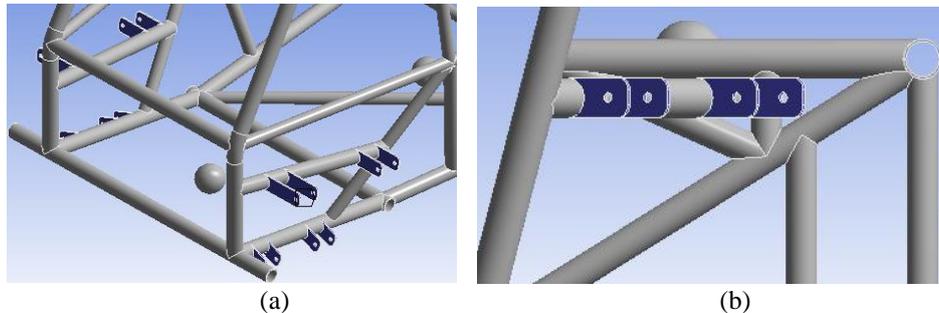

(a)   (b)

Figure 7. Bezel chassis points and representation of the sprung mass

Source: Author

Figures 8 and 9 show the form that loads will be applied in the pipes, in the case of frontal and lateral tipping rollover.
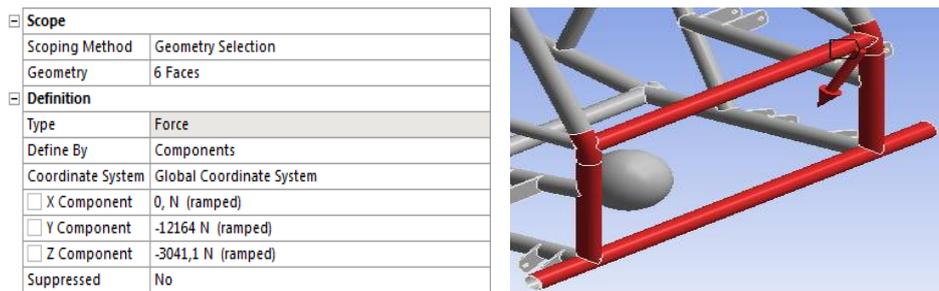

Scope	
Scoping Method	Geometry Selection
Geometry	6 Faces
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
☐ X Component	0, N (ramped)
☐ Y Component	-12164 N (ramped)
☐ Z Component	-3041,1 N (ramped)
Suppressed	No

Figure 8. Loads applied in case of front rollover

Scope	
Scoping Method	Geometry Selection
Geometry	5 Faces
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
☐ X Component	-3041,1 N (ramped)
☐ Y Component	-12164 N (ramped)
☐ Z Component	-4561,6 N (ramped)
Suppressed	No

Figura 9. Carregamentos aplicados em caso de tombamento lateral

2.4   Post-Processing

The post processing is the last step in the process of structural analysis by finite elements. This step is where it will be analyzed the results obtained with the simulations. It was possible to obtain some results for improving the structure by comparing the model 2015 and 2016. The results will be discussed in the next topic, where you can observe the main differences between the two models and their respective improvements.

3. RESULTS AND DISCUSSION

In the course of the work arose some barriers that hindered the implementation of a more complete and precise work, such as the use of the software, limitations on the subject involving finite elements, indecipherable computer limitations software errors.

We can apply the method of structural analysis established by the Australian ADR 59 Protocol, however due some preprocessing step errors given by the software, it was not possible to analyze the behavior of the structure on the basis of shipments.

Once again the dynamic analysis for (impacts and study of stiffness) was a challenge, due to your complexity. This work was performed only modal analysis, studies to assess the natural frequency of the structure. However, this study is extremely important, it's how we can avoid some failures from high frequencies. Each vibrate mode presents a deformation from a frequency any, assuming it's frequency has the same amplitude of the frequency generated by the engine for example, the framework can resonate and came to fail.

Modal analysis of free body were applied twelve modes of vibration, to be able to evaluate the structure in front of your geometry, knowing that for a geometry modelled correctly the first six modes of vibration features for frequency translation and its deformation must be equal to zero, and the last six rotation. The frequencies versus the vibration modes of the two models are presented in Figure 10 and 11.
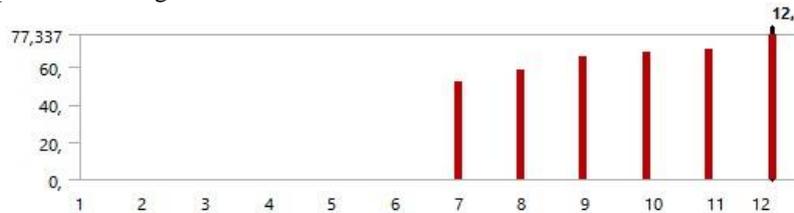

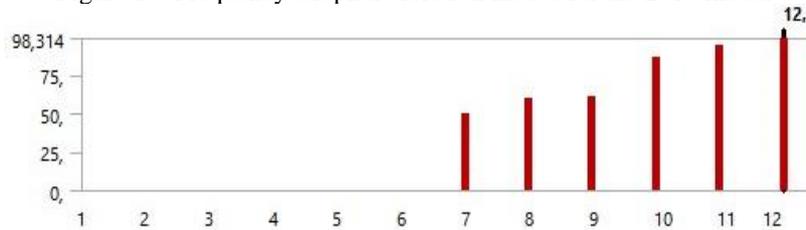Figure 10. Frequency Graph x vibration modes for the 2015 model

Figure 11. Chart x frequency vibration modes for the 2016 model

Watching the pictures 12 (A) and (B) we perceive directly that the crashes added to the 2016 model were efficient by reducing the deformations by rotation of the chassis using the same modes of vibration.
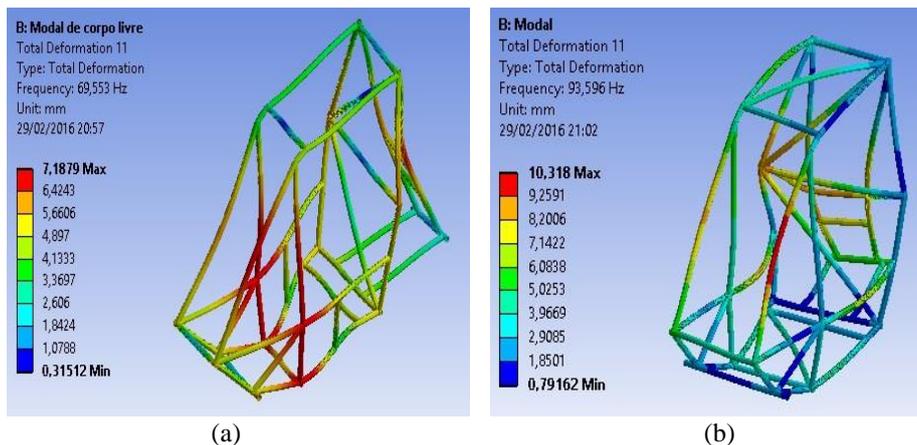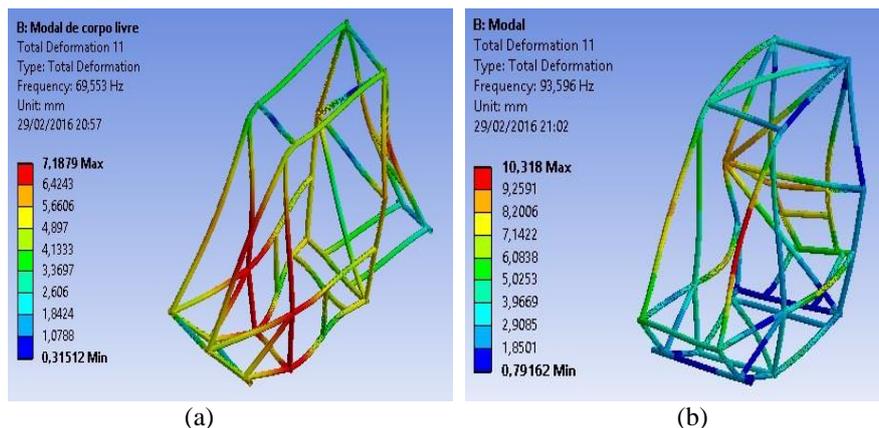

(a)   (b)

Figure 12. Represents free deformation body modal analysis by rotation (A) model
2015 without locking, (B) 2016 model with locking
Source: Author

In the analysis of rigid body with bound glass chassis, the model 2015 presented a frequency greater than the 2016 model. However the 2016 model features a maximum deformation greater than the previous

(a)   (b)

Figure 13. Represents modal analysis of deformation for rigid body rotation (A) model
2015 without locking, (B) 2016 model with locking
Source: Author

4. CONCLUSÃO

The development of this work helped to better understand the behavior of a tubular structure on the basis of their requests, especially the tubular chassis of a BAJA.

It was also possible to understand better what is the finite element method your importance in engineering in the development and optimization of new equipment, parts and structures in a way. The course of study was possible to notice that without specific equipment such as accelerometers and extensômetros not exist conditions of obtaining data (efforts) existing in real structure.

With adequate facilities, more in-depth knowledge on the subject it is possible to develop the work complete with real and accurate data. Finally, the 2016 model for being a lighter structure, we can increase your stiffness and consequently, making it a safer framework for the pilot.

5. ACKNOWLEDGEMENTS

I thank God first for having granted me the opportunity to do this work, I am also grateful to the IFPE-Caruaru Campus for all technical support, and my Advisor Diniz Branches, throughout your patience and companionship for the development of this work.

6. REFERENCES

Alves Filho, Avelino 1957- Elementos Finitos: a base da tecnologia CAE/análise dinâmica/ Avelino Alves Filho. –São Paulo: Érica,2005.

Alves Filho, Avelino 1957- Elementos Finitos: a base da tecnologia CAE/ Avelino Alves Filho. –São Paulo: Érica,2000.

Bizarro de Meireles, J. F. Análise dinâmica de estruturas por modelos de elementos finitos identificados experimentalmente. Universidade do Minho. Guimarães, 2007.

Manual de Aços, Gerdau Aços finos Piratini edição atualizada 2003

Fish, J,; Belytschko, T. Um primeiro curso em elementos finitos – São Paulo, LTC, 2009.

Freitas Fogal, M. L. Método dos Elementos Finitos aplicado em Análise Estrutural Estática. UNESPE – Campus de Ilha Solteira, 2002.

Furtado, Daniel Canongia. Análise Estrutural de Chassi de Veículos Automotivos / Daniel Canongia Furtado. Brasília: UnB, 2013.

João Pedro Taborda Lottermann. Projeto Estrutural de Chassi tipo Gaiola para Veículos Minibaja: Análise Numérica e Experimental / João Pedro Taborda Lottermann. – Brasília, DF, 2014.

Moura, H. G. "Métodos de Identificação de Forças em Sistemas Vibroacústicos", Dissertação para obtenção de grau de Doutor em engenharia mecânica, UFSC, 2010.

Norton, Robert L. Projeto de máquinas [recurso eletrônico] : uma abordagem integrada / Robert L. Norton ; [tradução: Konstantinos Dimitriou Stavropoulos ... et al.]. – 4. ed. – Dados eletrônicos. – Porto Alegre: Bookman, 2013.

Rao, Singiresu S. Vibrações Mecânicas / Singiresu S. Rao; revisor técnico José Juliano de Lima Júnior; tradução Arlete Simille – São Paulo, Pearson Prentice Hall, 2008.

SAE BRASIL, (Sociedade dos Engenheiros da mobilidade) <http://www.saebrasil.org.br/eventos/programas_estudantis/arquivos/Baja_2013_RBSB_7_-_Requisitos_Minimos_de_Seguranca_-_Emenda_3.pdf >, acesso em 20/02/2016.

7. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.