INTERNATIONAL JOURNAL OF ADVANCE RESEARCH, IDEAS AND INNOVATIONS IN TECHNOLOGY

ISSN: 2454-132X
Impact Factor: 6.078
(Volume 9, Issue 5 - V9I5-1147)
Available online at: https://www.ijariit.com

Free-body modal analysis of a Baja SAE vehicle chassis

Leandro de Paula Freire
leandrofreire.engmec@gmail.com
Instituto Federal de Educação, Ciência e Tecnologia de Minas Gerais, Arcos - MG, 35588-000, Brazil

Luiz Augusto Ferreira de Campos Viana
luiz.viana@ifmg.edu.br
Instituto Federal de Educação, Ciência e Tecnologia de Minas Gerais, Arcos - MG, 35588-000, Brazil

ABSTRACT

With the growing need to design increasingly efficient and complex systems, engineering studies are increasingly resorting to computer simulation techniques to analyze the performance and behavior of physical systems. These simulations help to reduce the time and cost of developing new projects. The aim of this article was to carry out a free-body modal simulation of the chassis of a Baja SAE off-road mini vehicle, using the finite element method. The study used Solid works software to generate the 3D model of the chassis and Ansys software to carry out the simulations. At the end of the simulations, it was possible to see that the chassis structure has natural frequencies between 36 and 86 Hertz (Hz) when the structure is free, which are different from the frequencies of the main source of forced vibration in the structure. In this way, it can be concluded that the structure does not enter the resonance phenomenon, meeting the design assumptions.

Keywords: *Finite elements. Modal analysis. Mechanical vibrations. Baja SAE. Chassis.*

1. INTRODUCTION

The Baja SAE program is an initiative that offers engineering students the opportunity to put into practice the knowledge acquired in the classroom. It is a challenge that aims to increase the preparation of students for the job market, providing valuable experience in the development of a mini off-road vehicle [1]. The Baja SAE program aims to challenge students to develop a detailed design, build and test a high-performance off-road vehicle, from conception to production. This gives students a broad view of the production and manufacturing processes, allowing them to get involved with a real product development case.

Vibrations in a vehicle's chassis are one of the problems that can affect vehicle comfort, safety, and stability. These vibrations can be caused by various factors, such as wheel misalignment, worn tires, problems in the transmission system or the engine. The study of vibrations in a vehicle chassis is one of the areas of interest in mechanical engineering, as it is necessary to understand the causes and consequences of these vibrations, thus being able to develop technical solutions to minimize or eliminate the problem. In addition, the analysis of vibrations in a vehicle chassis is fundamental for the design of new vehicles, to guarantee stability, safety, and comfort for the vehicle's occupants [2].

The aim of this study is to carry out computer simulations using the finite element method to analyze the natural frequencies of the Baja SAE mini vehicle chassis.

2. THE FINITE ELEMENT METHOD

Before the origin of computational tools, the behavior of a geometry subjected to stress was obtained by solving large systems of equations, making it impossible to obtain coherent results for complex geometries [3]. The finite element method (FEM) is based on the idea of simplifying problems that cannot be solved using analytical methods. The finite element method makes a major contribution to engineering and is indispensable in procedures related to the development of new products.

This tool is widely used to simulate the behavior of various components of a vehicle, helping to reduce production time and costs, since it is not necessary to have the car built to carry out tests. Solving problems with a high level of complexity requires the use of computers with a high level of performance. is important to simplify the problem and make it as close as possible to the real situation to be analyzed [2].

2. MECHANICAL VIBRATIONS

Mechanical vibrations refer to the oscillatory movement of an object or system around an equilibrium position. These vibrations are characterized by the periodic variation of the position, velocity, or acceleration of the object in question, in relation to time [4]. The frequency of mechanical vibrations is measured in Hertz (Hz) and represents the number of complete oscillations that occur in one second. The study of mechanical vibrations is important in various fields, such as engineering, physics, mechanics, and acoustics. It is possible to analyze the properties of mechanical vibrations, such as the natural frequency of an object or system, the response of the object to different forces, and even control vibrations to improve performance or prevent damage to equipment [5].

Vibration modes are the different ways in which an object can vibrate in a mechanical vibration. Each vibration mode is characterized by a natural frequency and a specific waveform, which is determined by the boundary conditions and the physical properties of the object [4].

3. MODAL ANALYSIS

Modal analysis is an engineering technique used to study the dynamic behavior of a system, usually a structural object or a machine, when subjected to mechanical vibrations. This technique makes it possible to determine the natural frequencies of the system, as well as the modal shapes associated with each natural frequency [6].

4. THE PHENOMENON OF RESONANCE

When the natural frequency is equal to the frequency of the forced vibration, the body enters a phenomenon known as resonance, which can cause large deflections in the structure [4]. To avoid the negative effects of resonance, it is important to identify the natural frequencies of systems and avoid applying external forces at these values or using damping techniques to dissipate the vibration energy.

5. CHASSIS RESTRICTIONS

The chassis structure of a Baja SAE vehicle must comply with the Baja SAE Technical and Administrative Regulations and must have primary and secondary members. The cage tubes must contain at least 0.18 % carbon and have an external diameter greater than 25.4 mm. In addition to the tubes mentioned above, it is also permitted to use structural tubes, which are generally used to attach powertrain components [1].

6. CHASSIS CONSTRUCTION MATERIAL

The chassis used in this study is made up of three different types of tubes. Fig-1 shows the chassis and the representation of the primary, secondary, and structural tubes.

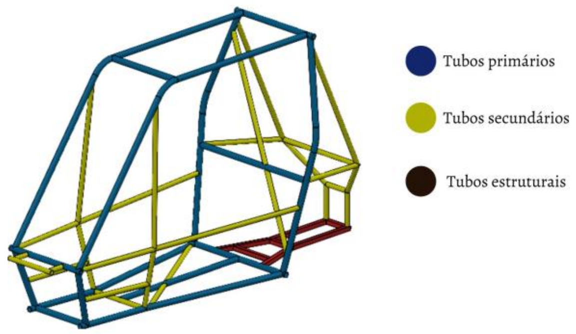

Fig-1 Chassis tubes

All the tubes in the chassis are made of SAE 1020 steel. Table-1 below shows the main geometric and mechanical characteristics of the tubes.

Table-1: Chassis tube properties and specifications

Modulus of elasticity	205 GPA
Tensile Yield Strength	351 MPa
Dimensions primary members	Ø 31.72 x 2 mm
Dimensions secondary members	Ø 25.4 x 2 mm
Structural profile dimensions	30 x30 x1.2 mm

7. STAGES OF THE STUDY

To carry out the free-body modal analysis of the model, it was necessary to follow a series of steps so that the simulation was as close as possible to the real situation. The stages developed in the work were developing the 3D model of the structure, preparing the geometry for the simulation environment, preparing the mesh, applying the contact conditions, and analyzing the results obtained.

8. 3D MODEL OF CHASSIS

The 3D modeling of the chassis was done using SOLIDWORKS 2021 software. The main aim of the chassis modeling process was to develop a design with a high safety index and the lowest possible mass, to make the vehicle more efficient. It is worth noting that all the chassis tubes have restrictions imposed by the RATBSB, which were considered in the modeling process. Fig-2 shows the modeling of the chassis used in the study.

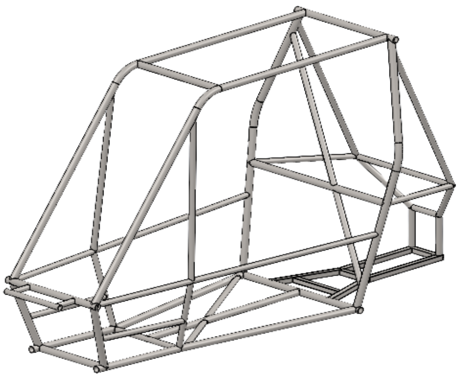

Fig-2 3D model of chassis

9. TREATMENT OF GEOMETRY

The 3D model of the chassis was imported into Ansys Software's Spaceclaim modeling environment, where the geometry was treated so that the program recognized the cage members as beam elements.

In finite element analysis, beam elements are one-dimensional elements defined by their length, cross-sectional area, and material properties [7]. They are used to represent the behavior of a beam or member that is subject to bending and shear stresses. This stage is of the utmost importance because it allows the characteristics of the tubes to be defined and the joints to be made. Fig-3 shows the geometry of the model prepared for the simulation environment.

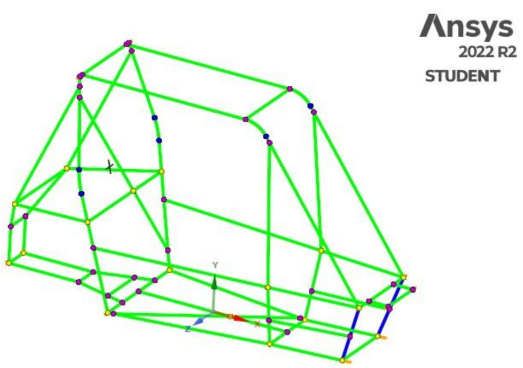

Fig-3 Treatment of geometry

10. MESH

The first step in the Ansys simulation environment is to generate the mesh for the study. The ideal mesh to be used for the model is found through a process known as mesh refinement, where the convergence of the results is checked. In this process, the mesh defined by the software is initially used and then refined to the point where the refinement does not cause major distortions in the results. Fig-4 shows the structure with the mesh defined after the convergence study, which is a 10 mm quadratic mesh with 3039 elements and 6012 nodes.

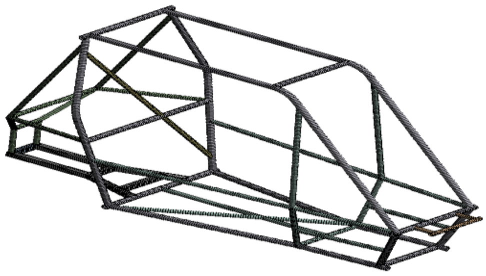

Fig-4 Mesh

11. CONDIÇÕES DE CONTORNO

The boundary conditions are essential for carrying out the analysis, as they allow the simulation to produce results that are consistent with the real situation. As the purpose of the study is to verify the natural frequencies of the structure, the boundary conditions will be defined based on the geometric characteristics of the chassis. Therefore, by defining the structural profiles and the material used, the software is automatically able to define the boundary conditions for the problem.

The biggest source of vibration in the chassis of a Baja SAE vehicle comes from the engine used in the car. As indicated by the regulations, the engine must be of the Briggs&Stratton brand and have 10 HP of power, which is shown in Fig-x. The frequencies for this engine model range from 15 Hz to 25 Hz, so from this data it will be possible to check whether the resonance phenomenon will occur in the structure.

Fig-5 Engine Briggs & Stratton 10 HP

12. MODAL FREE BODY ANALYSIS

After carrying out all the steps, it was possible to simulate the structure and obtain the natural frequencies of the first 6 vibration modes of the chassis. Table 2 below shows the free frequencies of the chassis.

Table-2 Natural frequencies of the chassis

Mode of vibration	Frequency (Hz)
1	39,755
2	41,707
3	60,189
4	75,898
5	78,018
6	85,268

Simulating the free-body modal analysis also makes it possible to check the direction in which the structure deforms. The only way to modify the structure's natural frequencies is to stiffen the chassis, so it is extremely important to check the direction in which the structure deforms. The following image shows the direction and size of the deformation displacement.

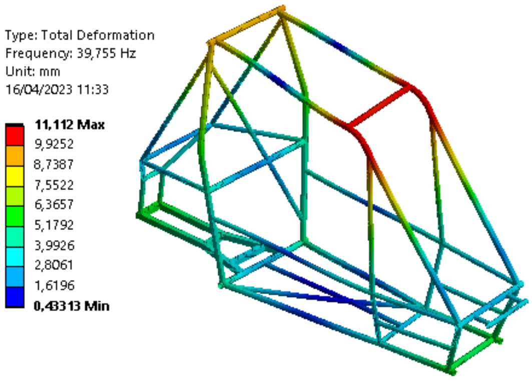

Fig-6 1st mode of vibration

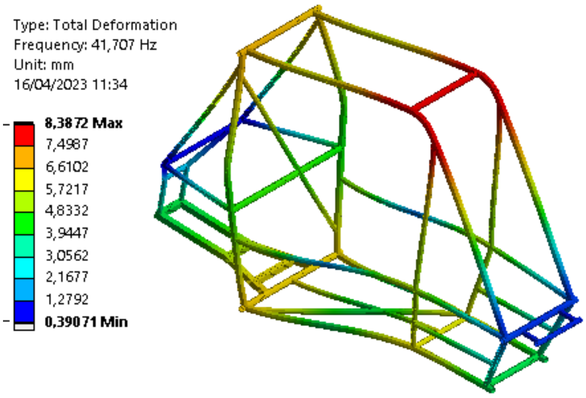

Fig-7 2nd mode of vibration

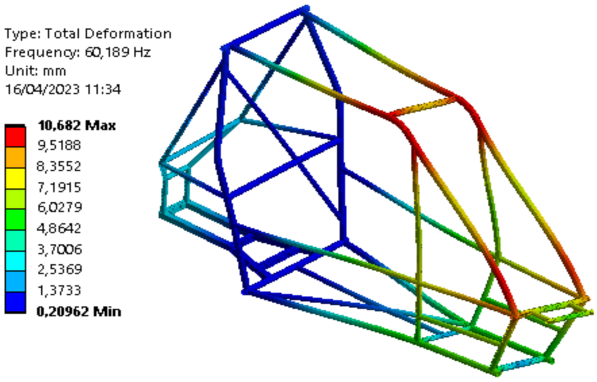

Fig-8 3rd mode of vibration

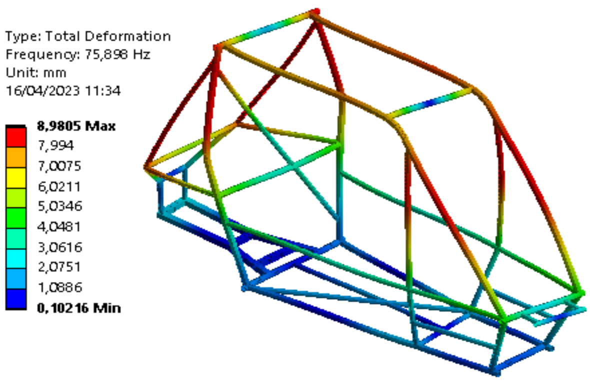

Fig-9 4th mode of vibration

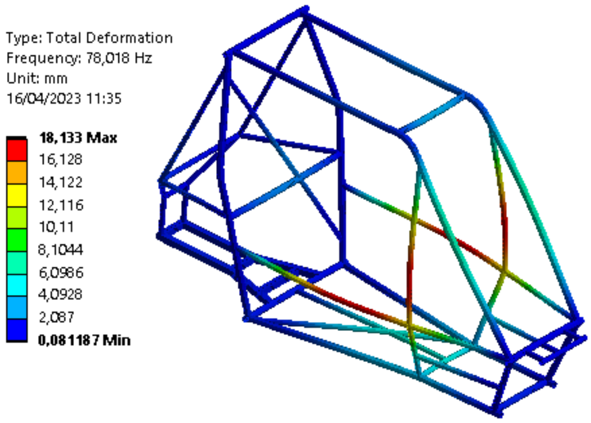

Fig-10 5th mode of vibration

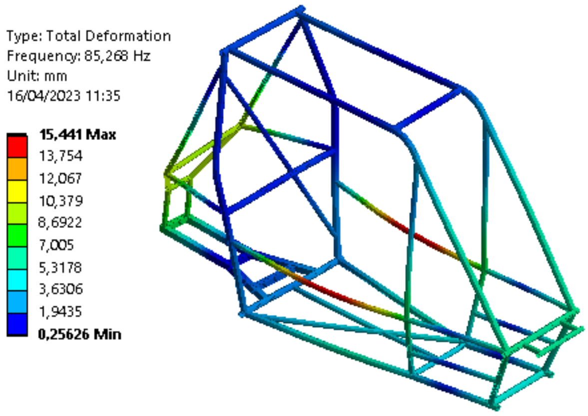

Fig-11 6th mode of vibration

Comparing the data obtained from the free-body modal simulation with the motor frequency, it is possible to see that the two do not have equal frequency values. Therefore, the natural vibrations of the structure have different values to the forced vibration source, ensuring that the structure will not be subject to the phenomenon of resonance.

13. CONCLUSION

In this work, the modal analysis of the chassis body of a Baja SAE vehicle was carried out. The calculations were carried out using the finite element method. For the study carried out, the geometric characteristics of the structure and the properties of the materials used in the chassis tubes were considered. The results obtained are consistent with expectations for a structure of this type, indicating that the structure is suitable for the application to which it will be subjected.

14. REFERENCES

[1] Rulebook 2023 - BAJA SAE International
[2] Gillespie, Thomas, ed. Fundamentals of vehicle dynamics. SAE international, 2021.
[3] Reddy, Junuthula Narasimha. Introduction to the finite element method. McGraw-Hill Education, 2019.
[4] Rao, Singiresu S. Mechanical vibrations. 2001.
[5] Meirovitch, Leonard. Fundamentals of vibrations. Waveland Press, 2010.
[6] Ewins, David J. Modal testing: theory, practice, and application. John Wiley & Sons, 2009.
[7] Belytschko, Ted, et al. Nonlinear finite elements for continua and structures. John wiley & sons, 2014.

