RESEARCH ARTICLE OPEN ACCESS

Linear Static and Dynamic Analysis of Rocket Engine Testing Bench Structure using the Finite Element Method

Luiza Fabrino Favato*, Professor Dr. Pedro Américo Almeida Magalhães Júnior**
*(Student of the Department of the Graduate Program in Mechanical Engineering, Pontifical Catholic University of Minas Gerais, Brazil)
**(Member of the Department of the Graduate Program in Mechanical Engineering, Pontifical Catholic University of Minas Gerais, Brazil)

ABSTRACT
This article presents a study of a testing bench structure for Rocket Engines, which is under development by the PUC-Minas Aerospace Research Group. The Bench is being built for civilian's liquid bipropellant rocket engines up to 5 kN of thrust. The purpose of this article is to evaluate the bench structure using the Finite Element Method (FEM), by structural linear static and dynamic analysis. Performed to predict the behavior of the structure to the requests of the tests. The virtual simulations were performed using a CAE software with the Nastran solver. The structure is 979 x 1638 mm by 2629 mm, consisting of folded-plates (¼ "x 3¼" x 8") and plates of 1/4" and 1/2 ", both SAE 1020 Steel .The rocket engine is fixed on the structure through a set called engine mount. It was included in the analysis clearances or misalignments that may occur during tests. As well as, the load applied was evaluated with components in varying orientations and directions. It was considered the maximum size of the engine mount and the maximum inclination angle of load. At the end of this article it was observed that the worst stress and displacement values obtained were for the hypothesis with the inclination of five-degrees with load components in the positive directions of the axes defined and it was also obtained the first twenty frequency modes of the structure.
Keywords – Dynamic analysis, Finite Element Method, Linear static analysis, Rocket engine, Test bench structure

I. INTRODUCTION

Test benches can boost research, projects and ideas. The development of new benches can encourage studies in several areas and consequently development and improvement of technologies. With increasing competition in the international market for space access, development, construction of rockets and associated components gain decisive importance in the context of a country's development. The Brazil only has the IAE's (Institute of Aeronautics and Space) test bench for rocket propellants, which is focused on the military sector, making it difficult for academics and privates tests. In an effort to expand the national aerospace activities the PUC - Minas Aerospace Research Group initiated a project to develop a Rocket Engine testing Bench, which mainly will allow the measurement of several parameters throughout the tests of rocket propellants prototypes. Recently the project was approved as a scientific initiation of FAPEMIG, the Foundation for Research of the State of Minas Gerais, APQ-01369-13 code. This article was developed to analyze the structural integrity of the bench structure, as well as to obtain its natural frequencies, using the finite element method.

I.I PROBLEM

The intention of this work was to developed a computer model that represents the impact of a test for liquid bipropellant rocket engine prototypes, up to 5 kN of thrust. In order to observe the capability of the structure to meet the demand safely. The problem was modeled to absorb possible faults, which may occur during the engine's test procedure. It was considered the unevenness of the propellant and the assembly motor's position on the structure through the component called the engine mount. The first natural frequency modes of the structure were calculated for future correlation with the motor's driving frequencies.

I.II HYPOTHESIS

The solutions presented in this article were obtained through considerations and hypothesis. The considerations were made due to load's orientation applied on the structure and boundary conditions. The hypothesis were presented with variation of load's magnitude and angle.

The refinement of the mesh was realized by analyzing the element in view of its size's modification and presented responses of stress and displacements. The assembly of the engine mount with the propellant was specified to have a maximum length of 560 mm. The load was applied in an area specified according to the area of a load cell, which has a diameter of 104.8 mm.

It was used a static linear structural simulation for three hypotheses of loading in relation to a possible inclination of the thrust caused by the mount's displacement. Another possibility explored was in order to obtain a curve variation of the maximum Von-Mises stress for a load variation until the worst loading condition for the structure. This hypothesis was evaluated by exploring the behavior of the structure to a possible fracture.

The modal analysis was performed considering boundary conditions, analyzing the degrees of freedom appropriated for the structure restrictions. It was considered that the first 20 natural frequencies are the most critical when it comes to a problem of the resonance phenomena. As the modes of vibration increase the energy required to occur resonance is higher. Thus, if the engine driving frequency is equal or closer to the natural frequency of the structure, at a very high frequency and there is not enough energy, the resonance phenomenon will not occur [1].

II. THEORETICAL BACKGROUND

The definition of rocket comes from Middle Ages which is a self-sufficient projectile, that carries its own supplies, the propellants: combustible and oxidant. The basic principle of a rocket is simple: matter or propellant is ejected in a high velocity, resulting in a force reaction, the thrust [2]. The rocket contains all propellants, being independent of the environment and can operate in space vacuum. Rockets can be classified by type (survey and satellite launch vehicles); propellant (solid, liquid and hybrid); number of stages (mono, multi-stage) and application (manned and unmanned) [3,4]. Moreover, rockets can have nuclear, chemical, and electrical motors.

A test bench for rocket engines must be designed and instrumented for determination of propellant's performance parameters. Some of them are engine's thrust values, propellant's flow rate, temperature and burning time. The bench is segmented into three main parts, which are mechanical structure, hydraulics and control systems [5].

III. METHODOLOGY

The adopted procedure began with literature review, and 3D drawing of the structure and engine mount. Then was determined the type of element and mesh, besides the 3D drawing was simplified creating a model shell, which it was more suitable for the analysis. In a CAE solver Nastran software the model was discretized and the generated mesh was refined. With the solutions offered by the considerations implemented, the model and the mesh went through a refinement process.

III.I GEOMETRICAL MODEL IN 3D

The 3D drawing of the structure of the Testing Bench for Rocket Engine of this article was developed on 3D CAD software. The structure has the maximum dimensions of 979 x 1638 mm by 2629 mm. It consists of folded-plates (¼"x 3 ¼" x 8") and sheets of 1/4" and 1/2 ".

It was also conducted the drawing of engine mount and laboratory's outline setup project (Fig.2), all developed by the PUC - Minas Aerospace Research Group. The upper part of the mount has degrees of freedom to allow the axial movement of the engine in order to push the load cell, which is responsible for collecting thrust values.

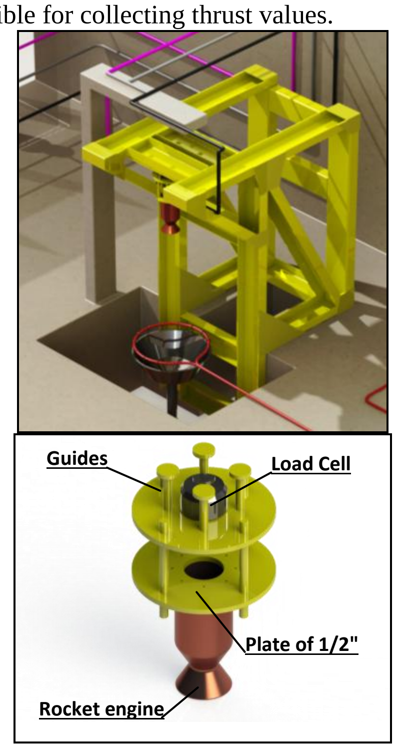


Figure 2 Outline of the rocket propellant laboratory and the engine mount

III.II COMPUTATIONAL METHODOLOGY

The computational methodology used was based on FEM, so it is segmented in determining the type of element, developing the mathematical model, executing the model discretization and analyzing the results.

The discretization of the model was accomplished by 1D elements, RBE2 and RBE3 and 2D elements, QUAD4 and TRI3, plate types, Fig.3. The fixing screws and the engine mount were represent by one-dimensional elements, RBE2 and RBE3, respectively. This selection was made because, the RBE2 is a one-dimensional element that transfers displacement and the RBE3 transfers force [6]. When one dimension, also called thickness, is much smaller than the other dimensions, the plate type element should be used. In accordance with this, the mathematical model of the Bench's structure was discretized using plate type elements. Moreover, a quadrilateral element is a more complete and refined element than the triangular element [7]. Thus, the main element used to discretize the model was the QUAD4. However, in almost negligible magnitude it was also used TRI3 elements due to geometrical difficulties Fig.3.

The finite element mesh was generated using NX Nastran, Fig.3. This figure represents the final mesh used for the analysis shown in this article, the maximum size of the element was 15 mm, and it was used 40620 QUAD4, 6 TRI3 and 11 1D elements. The total amount of elements was 41696.

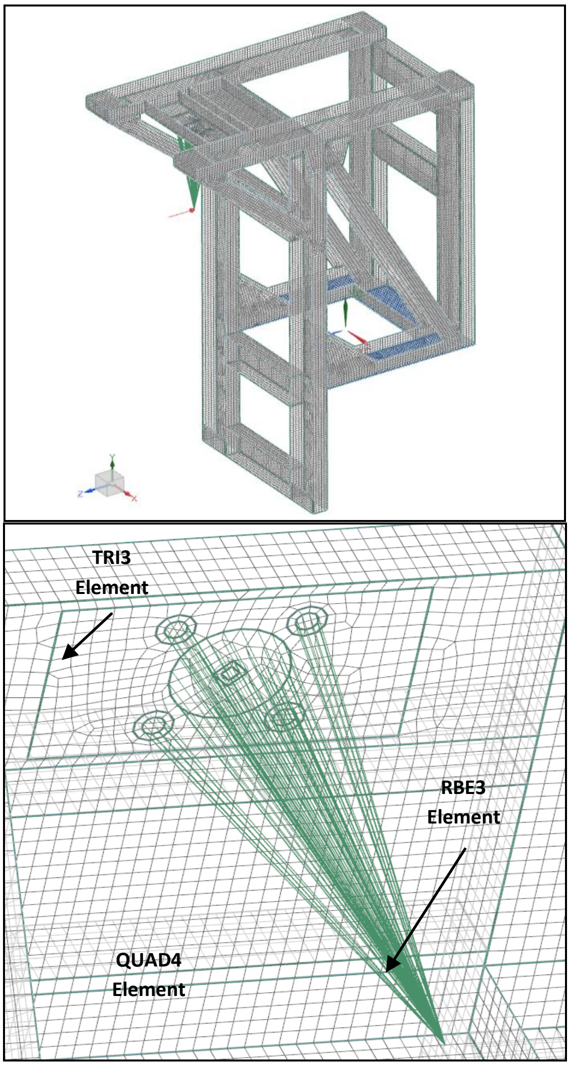


Figure 3 Isometric image of the generated mesh and detailed view of the use of 1D and 2D elements.

Before reaching this mesh, tests were performed and meshes were created by varying the size of elements (5, 10 and 15 mm). As the variation of stress values obtained was negligible and differences offset were decreasing closer to the element size of 15 mm, this element was chosen to discretize the model. Not only because of this, but also because the mesh generated by a larger element, has a lower number of elements and nodes than smaller elements, implying in a smaller numerical error in the results. The overall thickness of the mesh was 0.25 inches. However, some parts of the mesh were specified with different values of thickness because of the plates present in the structure.

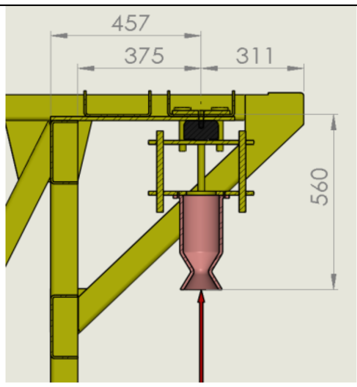
III.III LOAD HYPOTHESIS

In addition to the considerations it was assigned three testing engine operation hypotheses, due to hypothetical problems with the assembly, which may cause possible gaps or misalignments. In the hypothesis 0 it was established that during the test there would be no problems and misalignment with the bench structure and propellant mount assembly. Thus, the entire load was applied on the area of load cell. As for the other hypothesis, the load was applied at an angle to the planes YZ and XY. To facilitate the understanding of the assumptions Fig.4 was created.

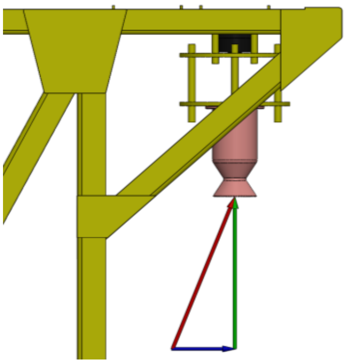
III.IV BOUNDARY CONDITIONS – RESTRICTIONS

In Fig.5, it is indicated where the boundary conditions were attributed to the model, which were added to it through the central node of the elements. The 1D RBE2 elements, in orange, represent the screws and the nodes displayed in blue assumed the role of the concrete.

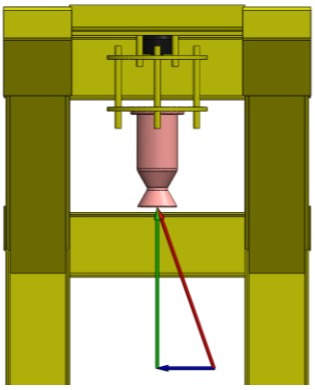
All degrees of freedom were removed of the elements in orange. In addition, on the elements in blue were added translation restrictions on the X,Y and Z axes and rotation restrictions on the X and Z axes.



Hypothesis 0 - load applied vertically



Hypothesis 1 - Load angled + Y and + Z



Hypothesis 2 - Load angled + Y and + X

Figure 4 - Hypothesis 0, 1 and 2

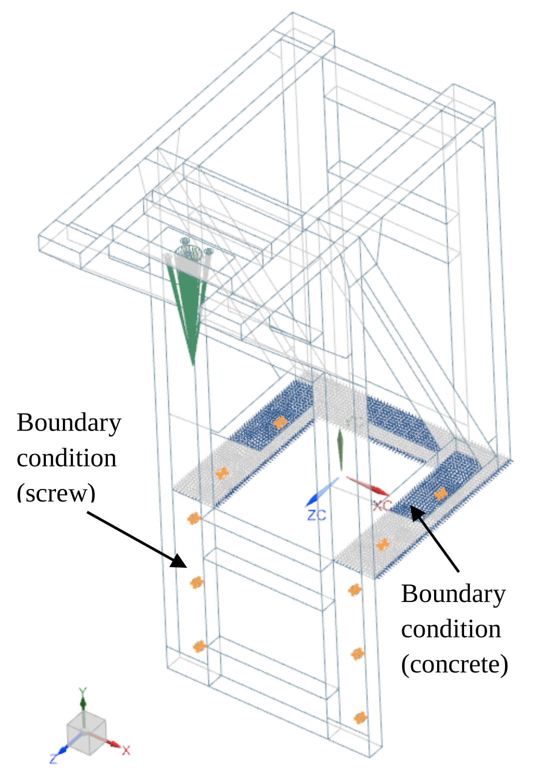


Figure 5 - Restriction assigned to the model

IV. RESULTS

It was initially analyzed the test results of the mesh and the mesh results characteristics. Then the results of the hypotheses were analyzed for the applied load of 5 kN in three load angle variations. Subsequently, the results of the modal analysis were evaluated.

It is illustrated in Fig.6 the mesh details for von-Mises stress result of the of hypothesis 1 with applied load of 5 kN angled in five degrees. Analyzing this mesh analytically the most critical area were in line with the application of load on the structure in the direction and orientation analyzed.

The results of displacements, minimum and maximum principal stresses and von-Mises stress for the critical hypothesis are shown in Fig.7. The overall results of the linear static structural analysis are shown in Fig. 8 for an applied load of 5 kN. On the 1 and 2 hypotheses the load was applied with 1, 3 and 5 degrees.

On Fig.8, it can be seen that the variation of the displacement values hanged from 0.169 to 0.190 mm. The minimum principal stress trend curves presented a variation from 9.887 to 13.90 MPa. The values obtained of the maximum principal stress were from 15.56 to 20.39 MPa. The range of values obtained for the Von-Mises stress were very close to those obtained for the maximum principal stress. The lower value of the Von-Mises stress presented was 16.25 MPa and the highest was 20.43 MPa.

The critical hypothesis was the hypothesis 1 at an angle of 5°, which presented the largest displacement value observed of 0.190 mm and the main minimum, maximum and von-Mises stresses of 13.90 MPa, 20.39 MPa and 20.43 MPa respectively.

The critical hypothesis was also analyzed through the increasing of the subjected load, 5, 41 and 50 kN and it was generated the load variation curve as a function of the von-Mises stress exposed on Fig.9

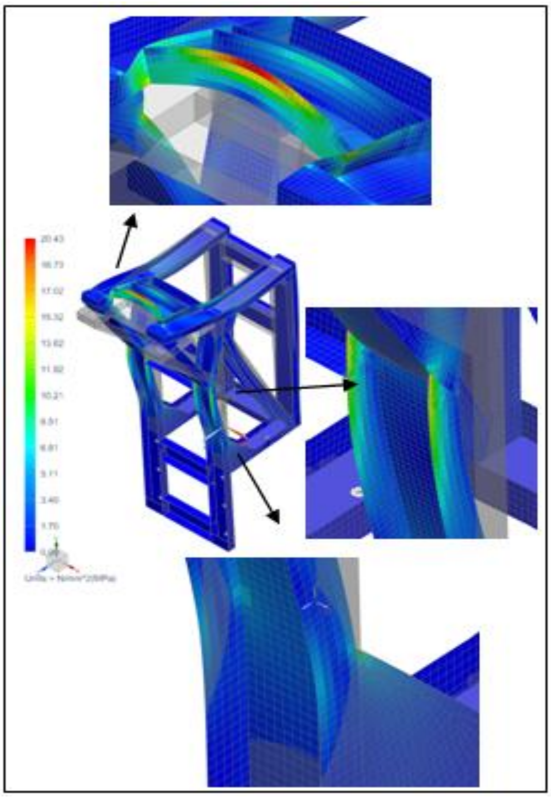


Figure 6 - Mesh details (Hypothesis 1-5 degrees - 5 kN - Von Mises - Structural linear static analysis)

It was also obtained the first 20 modes of natural frequencies of the structure in Hz through the modal analysis, Fig.10. The first mode frequency was 50.71 Hz and the 20th was 270.5 Hz.

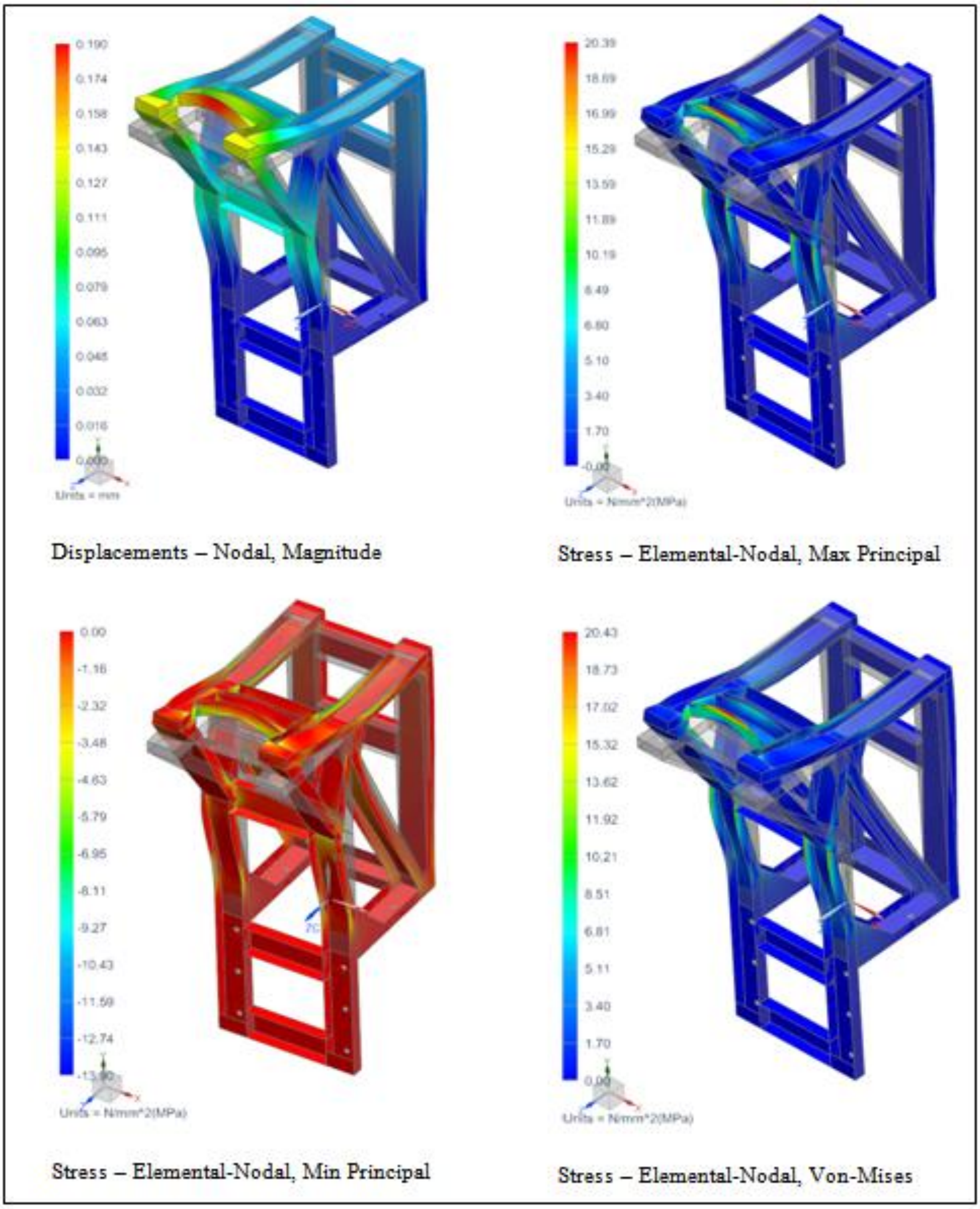
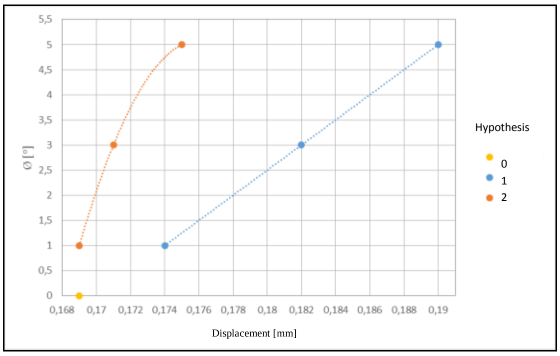
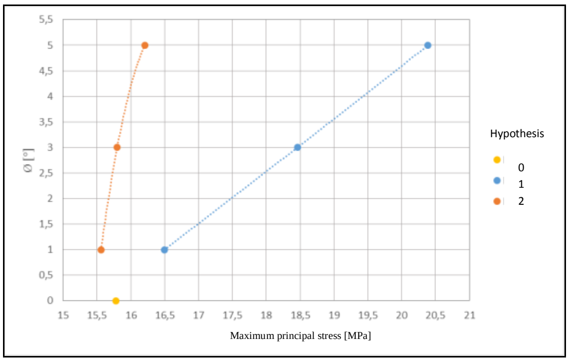


Figure 7 – Results of Hypothesis 1 - 5 kN Load – 5°

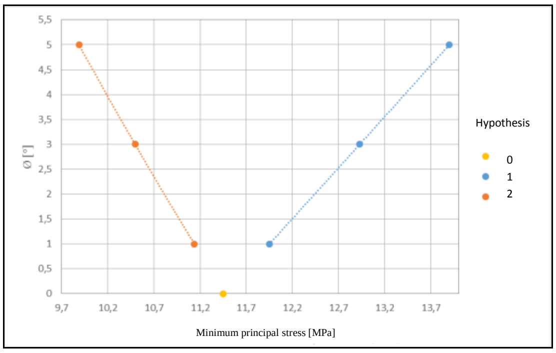
The results of the dynamic analysis are shown on Fig.10 and 11. It was possible to observe as expected a steady increase of the natural frequency of the structure as it was increased the frequency modes until the twentieth mode, Fig.10. As it is not known the driving frequency of the rocket engines to be tested in the Testing Bench, it is still not possible to identify whether will may occur the resonance phenomenon [1]. The Fig.11 illustrates the displacements of the first four modes of natural frequencies of the structure in mm. In general, for most frequency modes, region in which the engine mount is attached has the largest displacement values.



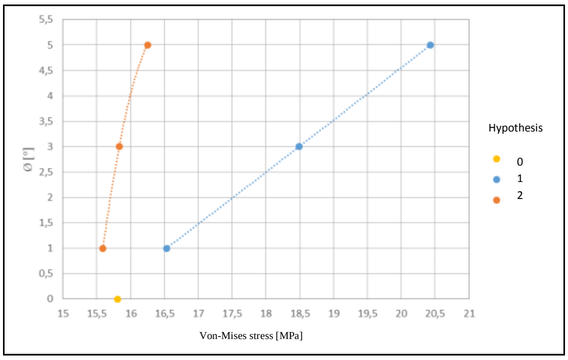
Maximum displacement X angle variation - Hypothesis 0, 1 and 2



Maximum principal stress X angle variation - Hypothesis 0, 1 and 2

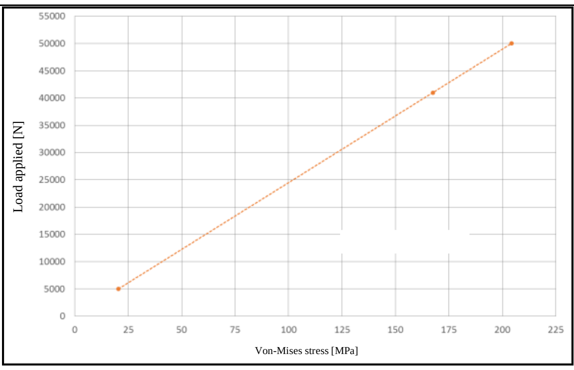


Minimum principal stress X angle variation - Hypothesis 0, 1 and 2



Von-Misses stress X angle variation - Hypothesis 0, 1 and 2

Figure 8 – Graphs of the overall results



Von-Mises stress X load variation - Hypothesis 1 – 5 degrees

Figure 9 - Graphic of von-Mises stress x variation's load for the critic situation

The results of the dynamic analysis are shown on Fig.10 and 11. It was possible to observe as expected a steady increase of the natural frequency of the structure as it was increased the frequency modes until the twentieth mode, Fig.10. As it is not known the driving frequency of the rocket engines to be tested in the Testing Bench, it is still not possible to identify whether will may occur the resonance phenomenon [1]. The Fig.11 illustrates the displacements of the first four modes of natural frequencies of the structure in mm. In general, for most frequency modes, region in which the engine mount is attached has the largest displacement values.

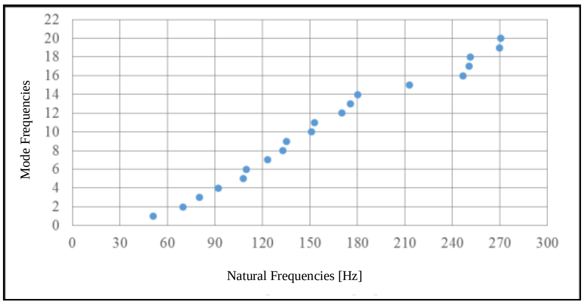
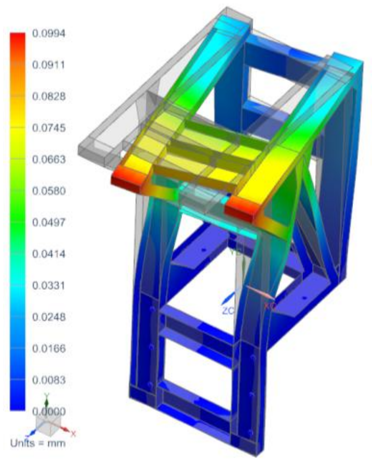
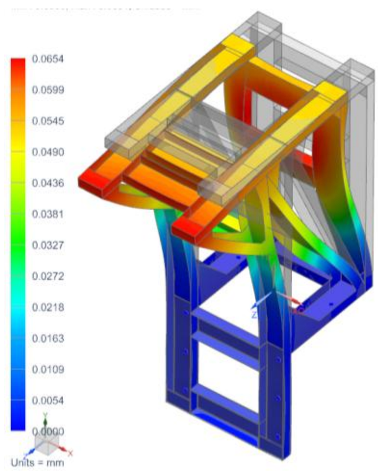


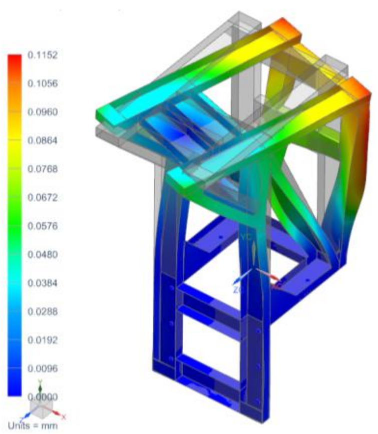
Figure 10 - Frequency Modes X Natural frequency



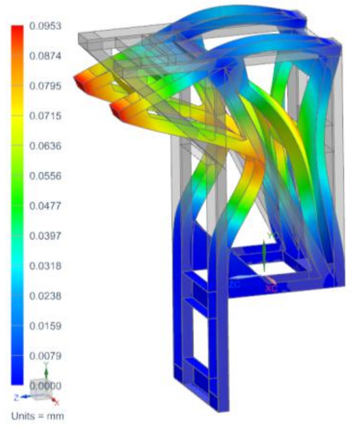
1st Mode - 50,795 Hz
Displacement – Nodal, Magnitude



2nd Mode - 69,921 Hz
Displacement – Nodal, Magnitude



3rd Mode – 80,27 Hz
Displacement – Nodal, Magnitude



4th Mode – 92,05 Hz
Displacement – Nodal, Magnitude

Figure 11 - Modal analysis results: 1st, 2nd, 3rd and 4th modes respectively.

V. CONCLUSION

It was possible to contextualize aerospace aspects and to presents the main features of a rocket engine testing bench. As noted by the results and by the assumptions made, it will not be necessary to make changes in the Testing Bench structure. As the project is being developed to support the tests of rocket propellants prototypes up to five kN. The maximum stresses presented in the analysis were very small, the biggest displayed value was 20.43 MPa, which is 9.7% of the yield strength of the SAE 1020 material, and the displacements ranged from 0.169 to 0.190 mm. Thus, the safety factor of the structure is 10. It was also obtained the first twenty modes of natural frequencies of the structure that will be important for future studies when the driving frequencies of the engine prototypes that will be tested were known. Also as a proposal for future work would be important to conduct a fatigue analysis on the structure. Thus, the objectives proposed for this work were achieved.

VI. ACKNOWLEDGEMENTS

The authors sincerely acknowledge the support, contributions and facilities extended to the survey and the article’s development by Vitor Thasso Ferraz Rodrigues; the staff and faculty of the Pontifical University Catholic of Minas Gerais, especially to professor Dr. Janes Landré Junior and professor Me. Athos Obvioslo Carvalho. Besides the teachers and members of the PUC - Minas Aerospace Research Group, especially to the professor Me. Welerson Romaniello de Freitas and FAPEMIG, the Foundation for Research of the State of Minas Gerais for the support and contributions related to the survey and the article.

REFERENCES

[1] D.J. Inman, *Engineering vibration* (Upper Saddle River: Prentice-Hall, 2008).

[2] A.B. Van riper, *Rockets and Missiles: The Life Story of a Technology Theory* (Westport, Connecticut, London: Greenwood Technographies, 2004).

[3] S. Nogueira, *Astronáutica: ensino fundamental e médio: Coleção explorando o ensino* (Brasília, 2009).

[4] J.W. Cornelisse, H. F. R. Schöyer, K. F. Wakker, *Rocket Propulsion and Spaceflight Dynamics.* (Pitman publishing limited, London, 1979).

[5] R.A. Contaifer, *Desenvolvimento de uma Bancada de Testes de Propulsores Híbridos de Baixo Empuxo*, máster teses, São José dos Campos, Instituto Nacional de Pesquisas Espaciais, INPE, 2009.

[6] NX Nastran. *Basic Dynamic Analysis, User's Guide*, 2008.

[7] H.L.S.L.S. Soriano, *Método de Elementos Finitos em Análise de Estruturas*. (São Paulo: Editora da Universidade de São Paulo; 2003).