CILAMCE 2017
Florianópolis - Santa Catarina - Brazil

MODAL ANALYSIS OF GUARANI I EXPERIMENTAL ROCKET

K. Máxima

P. G. C. Martins

S. H. S. Carneiro

kesianymaxima@gmail.com

paulomartins92@gmail.com

shscarneiro@unb.br

University of Brasilia

72430-201, Brasilia, DF, Brazil

Abstract. *During a flight mission, the rocket is in a harsh environment with aerodynamic loads due to velocity itself. These loads and the motor propulsion excite the structure of the rocket. The problem is that some frequencies of vibration can damage the rocket due to resonance. Modal analysis is used to avoid resonance into engineering problems. Therefore, it is important to know the vibration dynamics of a rocket in order to better define the place of certain equipment and sensors inside it and to avoid structural damages. This article deals with a modal analysis of GUARANI 1 experimental rocket. Some numerical simulations were performed in order to get the natural frequencies and vibration modes. These simulations were realized on the software Modal available on ANSYS and in the Engineering vibration toolbox for MATLAB (Slater et & Timbo, 2015). For the simulations two cases were considered: the first one with the complete rocket, accounting for the propellant, fins, nose, nozzle, avionics and valves, and the second one with only the main structural components of the rocket. Both of the simulations considered the rocket fixed on the Launchpad. The simulation on Engineering vibration toolbox were realized by using the Timoshenko 2D Beam Theory developed by Stephen Timoshenko (1921). To simulate on ANSYS, the mesh were simplified by using different element types to accommodate well a mesh grid over all the rocket. As results from ANSYS we have that among the 150 first modal shapes, almost 60% of them were vibration modes of the stringers, and 23% were bending modes of the rocket. Moreover, some modes of vibration of the fins, coif and torsion modes could be observed, these ones couldnt be seen on the MATLAB results since it was only a 2D simulation.*

Keywords: *Modal Analysis, GUARANI I, Rocket vibration, ANSYS Modal, vibration toolbox*

1 INTRODUCTION

One of the most critical parts of a rocket design is to predict its dynamic behavior during the flight. Vibration during the flight can affect some very important systems, such as the control system, the structure integrity and the pressurization system, as well as the payload. For example, the communication hardware installed in the last stage during the mission, or some release of structural part due to the vibration that could cause a disaster (Gierse et al., 2013). One of the Apollo missions had the first stage committed by vibrations from the combustion, where longitudinal oscillations caused the "Pogo effect" and, consequently, the main engine shut down (Sauser, 2009).

This work presents a numerical study of the dynamic behavior of the experimental rocket GUARANI 1, and have the objective of determining numerically the first modes of vibration and the natural frequencies. Moreover, it has been made a study of the response to an excitation with magnitude of a thousand Newtons at the nose of the rocket on the longitudinal direction.

The GUARANI 1 experimental rocket was developed and constructed by students from the major aerospace engineering at University of Brasilia, with the objective of participating on the Intercollegiate Rocket Engineering Competition - IREC. Its mission is to reach an apogee of 3km and to release a payload. It has an hybrid motor and a control system.

2 LITERATURE REVIEW

The modal analysis is an important tool used to determine vibration modes, natural frequencies and damping factors in structures study. It has a very simple implementation and in an experimental test, it can be not destructive. This analysis determines the dynamic properties and responses of a system submitted to vibration excitations (Leme, 2014).

The numerical method used in this study for the modal analysis of the experimental Guarani I Rocket is the FEM - Finite Element Method, which can be used to solve complex problems by replacing it by a simpler one, where the solution region is formed by interconnected finite elements. This numerical method have been more widely used for solving structural mechanics problems, but it has been successfully applied to heat conduction, fluid dynamics and others engineering problems (Rao, 2011).

The simulation executed by using Matlab used the Finite Element Method with only Timoshenko Beam elements. The Timoshenko Beam Theory is a good method used when the object in study has one really long dimension in comparison to the others, like a long beam (Inman, 2014). In comparison to the theory of Bernoulli for beam elements that ignores the effects of shear deformation and rotary inertia, the theory of Timoshenko calculates them and, consequently, it has a less stiff behavior than a Bernoulli beam model (Nelson & Wang).

In order to apply the beam theory one very important assumption must be made: that the cross-section of the beam translates and rotates as a rigid body, remaining plane and keeping its shape during deformation (Andersen & Nielsen, 2008).

ANSYS Modal allows others element types besides beam elements to be used on the numerical study. It allows to use shell and solid elements too, which gives more flexibility to generate a mesh, and can better represent a multi-shape structure. But, it must be taken some care when using different element types in the same mesh, since the incorrect coupling of different element types can model a rigid body motion (Nelson & Wang).

3 METHODOLOGY

For the numerical simulations, the rocket was analyzed at two different configurations, the complete rocket, weighting 24.67 kg, 2.52 high and with a diameter of 0.15m, and the simple rocket, with only structural components and weighting 10.71 kg, presented in the figure 1. Both configurations have the rocket longitudinally placed on the y-axis.

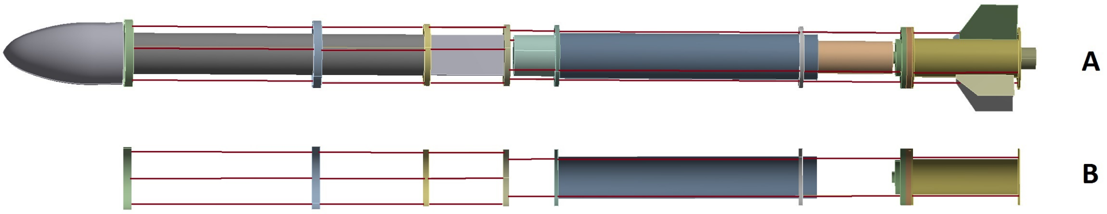

Figure 1: A - complete rocket, B - simple rocket

3.1 Matlab Numerical Analysis

For the Matlab simulation, the Engineering vibration toolbox 8 (Slater & Timbo, 2015) was used, which is related to the Finite element Method.

To input the geometry, boundary conditions and material properties was utilized the example available on the folder of installation from this program. This rocket were treated just like a cantilever beam fixed on one extremity. The parameters required on this software are the coordinates of the nodes, the material properties of each element, like the linear density, Youngs modulus, cross section area and moment of area of the structural part. Moreover it was necessary to include the effect from the non structural part of the rocket, like the fins, nose, fuel, oxidant, coif, valves and avionic. They were considered as concentrated mass on each node.

The boundary conditions considered were the fixing of the bottom of the rocket and free on all other nodes, to simulate the complete rocket installed on the Launchpad, or set up of the simple rocket over a shaker.

After all this process, it were acquired the modal shape and natural frequencies of the two cases considered, because the difference between the configuration of the simple rocket for the complete one, is the non consideration of the concentrated mass of the non structural material over the nodes, the rest remained the same.

3.2 Modal ANSYS Numerical Analysis

For the numerical analysis on ANSYS, SpaceClaim was used in order to simplify the geometry. For this simplification, the rocket mesh was constructed by using three different element types:

- Beam elements: elements which have only one dimension, they could be well applied to the stringers of the rocket since these stringers are very long and have a small transversal area.
- Shell elements: two dimensional elements applied to the fins and nose of the rocket, since these component are very thin in one dimension.

- Solid elements: three dimensional elements applied to the motor, tank, valves, avionic, parachutes and the rocket fuselage.

After this simplification, the mesh grid were set up over all the structure presented on figure 2. The following mesh have 20922 elements and 74757 nodes for the complete rocket and 9880 elements and 34709 nodes for the simple rocket. The methods used to realize the mesh for the simple rocket were the SOLID186, BEAM188, CONTA174 and MPC184, the complete rocket beyond those methods it used the SHELL181, SOLID187, CONTA175 and TARGE170.

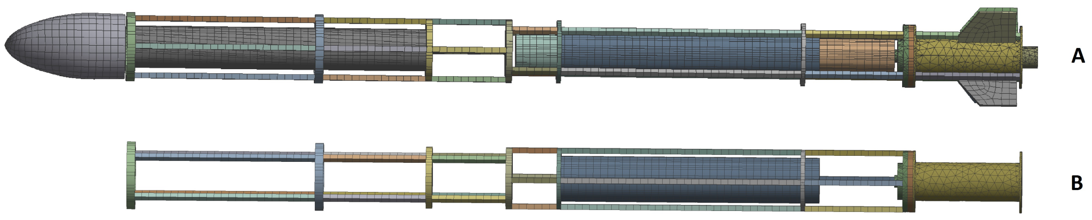

Figure 2: A - Mesh over the complete rocket, B - Mesh over the simple rocket.

After the configuration of the mesh and the simplification of the structure, it were set up the boundary condition, fixing the bottom of the rocket, just like it were done on the Matlab simulation. Then it was obtained easily the modal shapes and natural frequencies for both cases.

4 RESULTS AND DISCUSSION

The result from ANSYS and Matlab simulations are showed in table 1 below where the first five bending and longitudinal natural frequencies for the same plane are compared. Since the ANSYS simulation is 3D, it was necessary to select only one plane of symmetry (yz plane) in order to compare the results to the 2D simulation on Matlab.

Table 1: Natural Frequencies and modal shapes.

Modal shape	Natural Frequency (Hz)					
-	Complete Rocket			Simple Rocket		
-	ANSYS	Matlab	Difference (%)	ANSYS	Matlab	Difference (%)
1 - bending	14.49	8.13	43.89	20.00	17.63	13.58
2 - bending	61.61	48.17	21.81	52.00	87.00	65.44
3 - bending	108.23	132.00	21.50	102.82	244.00	136.93
4 - bending	273.08	250.00	8.61	176.08	437.00	148.38
5 - bending	335.48	416.00	23.98	234.62	752.00	220.50
1 - longitudinal	353,17	220.00	37.82	463.51	437.00	5.68
2 - longitudinal	913.57	539.00	40.99	1039.08	939.00	9.64
3 - longitudinal	1264.90	991.00	21.66	1939.10	1703.00	12.16
4 - longitudinal	1895.10	1578.00	16.73	2545.70	2395.00	5.93
5 - longitudinal	2391.50	1920.00	19.71	3209.90	2692.00	16.14

For the comparison of the results showed in table 1, it is good to start from the longitudinal modes, since "*the longitudinal vibration is the simplest to analyze*" (Rao, 2007, pg. 234) and, therefore, the beam approximation used on Matlab for the entire rocket and the multi-elements mesh used on ANSYS are more likely to be compared.

The natural frequencies determined from Matlab for the longitudinal vibration of the complete rocket has assumed lower values than the ANSYS modal results for the same case. This results can be explained by the fact that the ANSYS simulation has a stiffer model because the non-structural components, like the propellants and avionics, contributes for the rigidity of the rocket, differently from the Matlab simulations were these components are represented only by concentrated masses and do not contribute to the overall stiffness of the rocket. So, because the natural frequencies are directly proportional to the stiffness, they are higher for the ANSYS numerical analysis.

To get less difference between the longitudinal natural frequencies of the rocket simulated on Matlab and ANSYS, the simple configuration of the rocket was studied. As results, ANSYS and Matlab numerical analysis presented more similar longitudinal natural frequencies.

All the bending natural frequencies on Matlab has a tendency to decrease the accuracy in comparison to the results obtained through the ANSYS Simulation. This is due to the quality of the mesh on Matlab, the different method used by the solvers, the inability to consider all the geometry features on Matlab and the distinct way to treat the stringers, in ANSYS they were treated as beam elements with constant cross section.

A good correspondence is noted by the Matlab simulation for the first modal shape. Due to it's lowest natural frequency the most important bending modal shape is the first one, with that, it may be acceptable to consider the Engineering vibration toolbox for very preliminary analysis of the resonance due to the complexity of the geometry and the short time for simulation studies. The first bending modal shape of vibration for both cases simulated on ANSYS and Matlab are showed on figure 3.

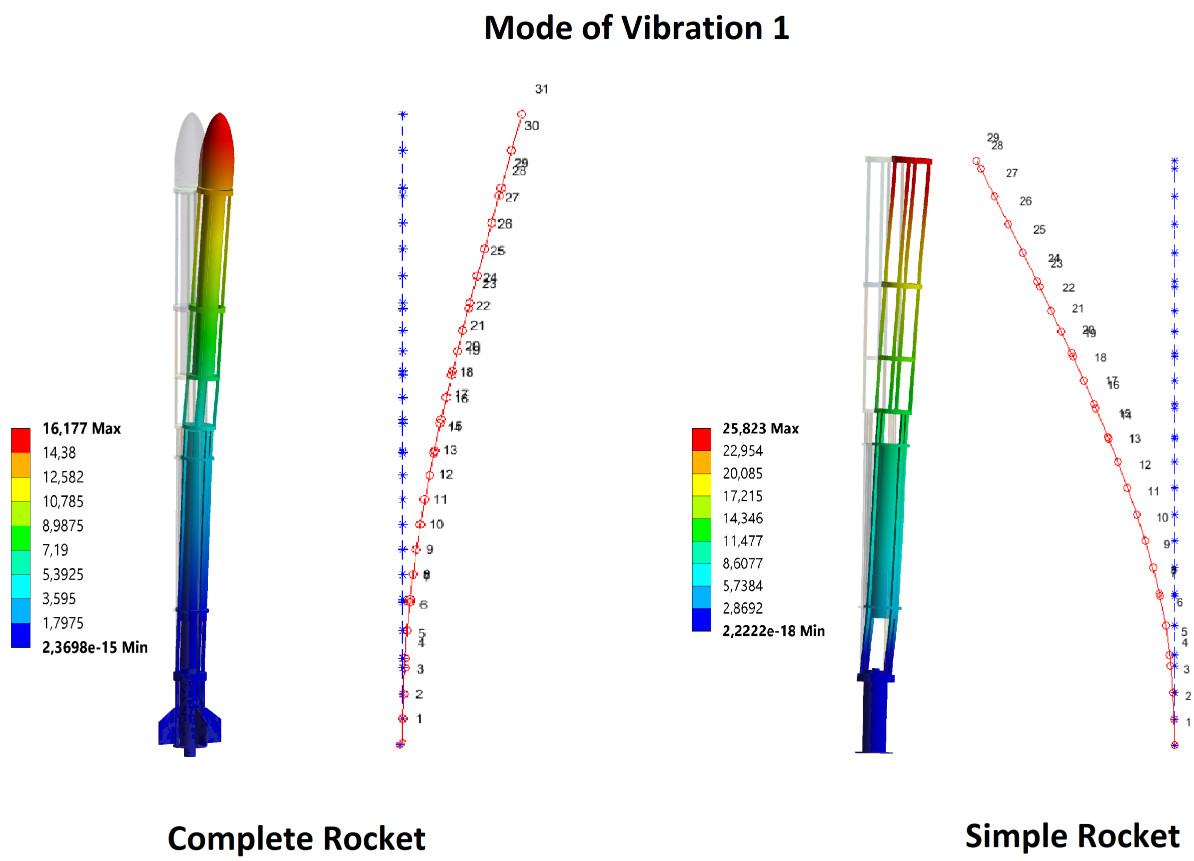

Figure 3: Modal Shape 1

The first bending modal shape are pretty similar for the complete and the simple rocket configurations and for the ANSYS and Matlab analysis, as shown in figure 3. However, the bending deflection showed to be higher for the simple rocket, with a small increase of the motor system inclination.

The simulation on ANSYS has many advantages in relation to the Matlab one. First, it allows the generation of a mesh with multi-types of elements and, then, it can be arranged to better represent the geometry. Second, it can be made on 3D, taking into consideration non-symmetric geometries and their results on different planes. As results, the ANSYS Modal analysis presented more than only the bending and longitudinal modal shapes of the rocket, as can be seen on figure 4.

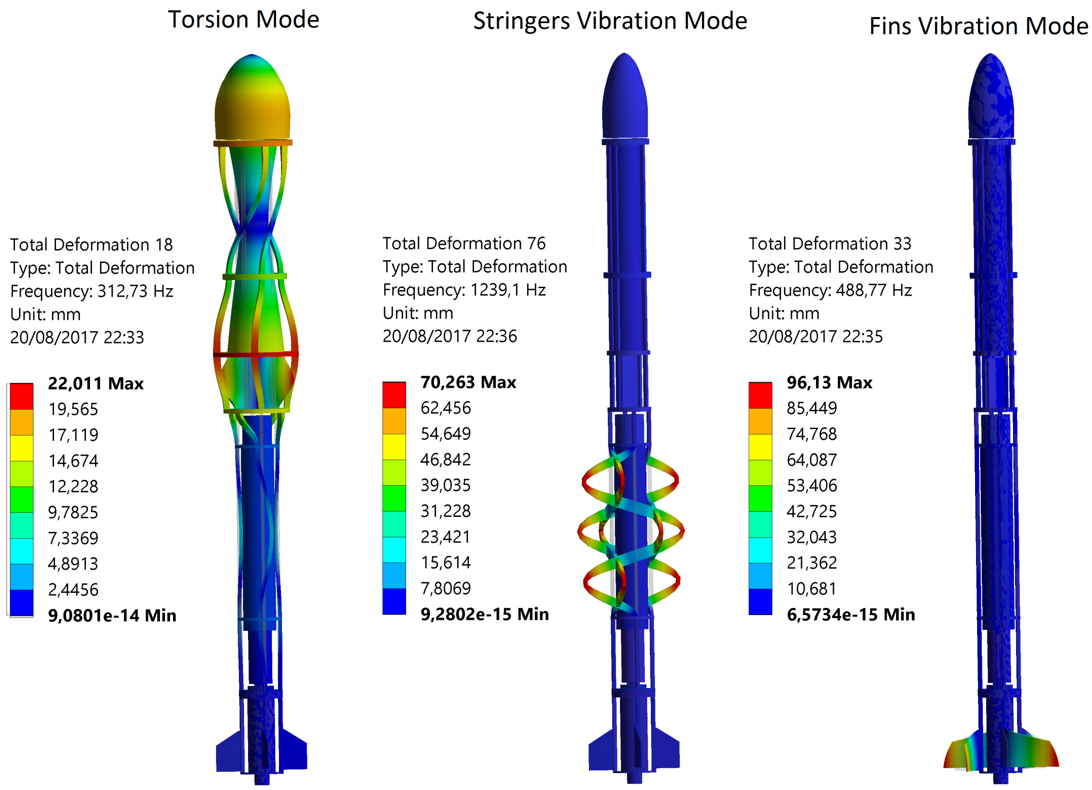

Figure 4: Torsion, stringers and fins modal shapes.

For the 150 first modal shapes analyzed on ANSYS for the complete rocket, 27 were bending, 5 longitudinal, and 7 torsional modes of the entire rocket, 89 were modes where only stringers vibrate, 12 where only the fins vibrate, 4 where only the nose vibrates, and 6 where only the avionics structure vibrates. It is important to say that these modes can present more than one vibration type, like bending and torsion, or the nose vibrating with some stringers bending, so they were identified for the most significant vibration observed.

The vibration modes of the fins are very important to analyze, because they must be made stiff enough to not fracture due to fatigue that can, for example, be caused by aerodynamic forces such as wind load. Other important vibration modes are the ones where only the avionics structure vibrates. Avionics data readings and processing, and wire junctions can be very sensitive to the rocket dynamics and must be placed where they can avoid intense movements. So, the results showed that the avionics of GUARANI I Experimental Rocket must receive special attention because a lot of modal shapes present the highest deformation on the avionics structure, in addition to the modes where only it vibrates. Then, it may be reinforced or change places in order to avoid failures of the avionic system and, consequently, failure of the data acquisition system, the payload ejection or the rocket recovery.

5 CONCLUSIONS

The present numerical modal analysis of GUARANI I Experimental Rocket is a start for the dynamic behavior study of this rocket, and must be yet validated by experimental tests. This study presented promising results that can be used to have an idea of the critical vibrations which can cause some system failure or structural fracture, in addition to notice the regions which suffer more from these vibrations. However, only experimental results will confirm this numerical analysis.

For the calculation of the longitudinal modal shapes and their natural frequencies, it was showed that the Engineering vibration toolbox can be used as a first approximation with a good agreement with the finite elements method. But, the bending results showed that they must be reformulated after some experimental analysis.

The ANSYS Modal presented very dissimilar bending natural frequencies and some modal shapes compared to the Matlab analysis. This showed to happen not only because of the rigidity contribution of the non-structural components of the rocket, but mainly because of the simulation parameters as the dimensions and element types considered, which do not showed to have so much effect on the longitudinal natural frequencies and shapes. Moreover, this 3D analysis presented some modal shapes that could not be observed on Matlab, like torsional and bending/torsional modes, in addition to some vibration modes of the stringers, fins and coif.

REFERENCES

Andersen, L., Nielsen, S. R. K., 2008. *Elastic Beams in Three Dimensions*. Aalborg University.

Gierse, A., et al., 2013. *Experimental In-Flight Modal-Analysis of a Sounding Rocket Structure*. ResearchGate.

Inman, D. J., 2014. *Engineering Vibration*. Pearson, Fourth Edition.

Leme, L. M. M., 2014. *Comparative Study of Experimental Modal Analysis Techniques in the Frequency Domain on a bar with the free-free boundary condition*. TCC-Mechanical Engineering, Pato Branco.

Nelson, T., & Wang, E. *Reliable FE-Modeling with ANSYS*. CADFEM GmbH, Munich, Germany.

Rao, S. S., 2007. *Vibration of Continuous Systems*. John Wiley & Sons, First Edition. Pg. 234.

Rao, S. S., 2011. *The Finite Element Method in Engineering*. Elsevier, Fifth Edition.

Sauser, B., 2009. *What's the Deal with Rocket Vibration?*. MIT Technology Review. Retrieved from *https://www.technologyreview.com/s/414364/whats-the-deal-with-rocket-vibrations/*.

Slater, J. C., & Timbo, R., 2015. *The Engineering Vibration Toolbox*. Software Package.

Timoshenko, S. P., 1921. *On the correction for shear of the differential equation for transverse vibrations of prismatic bars*. Philosophical Magazine, n. 41, p.744-746