XL CILAMCE
IBERO-LATIN AMERICAN CONGRESS ON COMPUTATIONAL METHODS IN ENGINEERING
NOVEMBER 11-14, 2019
Praiamar Natal Hotel & Convention
Natal, RN-BRAZIL

MODAL ANALYSIS IN A 2U CUBESAT STRUCTURE

Vitor H. L. C. Lima
Romulo do N. Rodrigues
Roberto de A. Bezerra
Pierre M. C. Lamary
vitorhugo_mec@alu.ufc.br
eng.romulorodrigues@gmail.com
roberto.bezerra@gmail.com
pierre.lamary@free.fr
Mechanical Engineering Department, Technology Center, Federal University of Ceará
Bloco 714, Centro de Tecnologia, Campus do Pici, Fortaleza/CE, CEP 60440-554, Brazil

Abstract. The miniaturization of satellites has helped students practice their knowledge and build a real satellite. Many universities have turned to this study, as other private and important institutions. The CubeSat standard gives a set of specifications to developers and defines a cubic picosatellite with less than 1.33kg. The vibration tests are some set of important tests to improve a satellite structure safety. Also, before doing vibration tests, FEM analysis is largely used to analysis the CAD model, before built the satellite. In this work is aimed make some modal analysis in an ordinary structure developed. The results can be applied in a real CubeSat that is going to be developed and later withstand the vibration tests in the Mechanical Vibration and Acoustics Laboratory of Federal University of Ceará. It was made modal analysis varying the load conditions and putting a circuit plate. In addition, it was also made harmonic analysis evaluating slack in the CubeSat's feet. Some changes in the mode shape were found in the presence of the circuit plate, and principally with slacking in the feet. There are mode shapes that all the deformation concentrates in the circuit plate, this natural frequency may damage the circuit. The results and observations will help in future analysis and tests.

Keywords: 2U cubesat, Modal analysis, Finite element method, Boundary conditions.

1 Introduction

The CubeSat standard has made a large development in space studies, because it is a cheap manner to launch payloads into space. As David [1] supports in his article, several Universities has researched the development of CubeSats as an opportunity to learn and practice about building a satellite. Also, the miniaturization makes possible a better acceptation of the failure since the cost is bankable and using a constellation of low-cost satellites ensures the mission be successful, said Quiroz-Garfias [2].

Not only universities are interest in this satellite model, but another institution like enterprises, military units and the NASA, as says Hansen (apud BÜRGER) [3]. Quiroz-Garfias [2] complements, the growing of small satellite industries has made this technology more sophisticated and more cost-effective.

In 1999, Prof. Puig-Suari and Prof. Twigs began to develop a standard design of picosatellite to help the access to space, so the CubeSat model was created. A CubeSat is a cubic structure with dimension of 10 cm of flank and mass of up to 1.33 kg. The capacity may vary, since the CubeSat can be built as two or more CubeSats piled up. California Polytechnic State University (Cal Poly) also provide a standard to CubeSat developers with a set of specification to build a CubeSat with all the safe conditions and the right design to be transported in a Poly Satellite Orbital Deployer (P-POD). The principal materials required for the standard is Aluminum 7075, 6061, 5005 and 5002. (CALYFORNIA POLYTECHNIC STATE UNIVERSITY [4])

Some series of tests should be performed in the satellite before being launched. Vibration, Thermal and Shock tests is dated by California Polytechnic State University [4], but the conditions of the test should respect all the requirements of the launch provider. The most common Vibration Test made in satellites are the sine-burst test, random vibration test, sine sweep test and sine-dwell test, and the main way of verifying compliance of all those tests is doing they in a shaker (SARAFIN *et al* [5]).

Finite Elements Method (FEM) also is a large way used in published works about vibration tests in satellites, Quiroz-Garfias [2], Bürger [6], De Aguiar [7], Chiranjeeve [8] show some different ways to do vibrations analysis using FEM. The modal analysis by FEM gives several information about the behavior of the structure in its natural frequency, when there is large dissipation of energy.

Recently, the Mechanical Vibration and Acoustics Laboratory (LMV) of Federal University of Ceará (UFC) began studies about vibration tests in CubeSats. Since the studies are in the beginning, was decided to start with FEM analysis, which includes harmonic, modal and random vibration analysis. So, in this work is wanted to do some primary analysis in a designed 2U CubeSat model (two cubes with 2.66 kg) to do more efficiently FEM analysis in a real CubeSat that is going to be built and do the vibration tests.

This work also aims to observe the effects of the vibration in a circuit plate and the influence of the circuit plate in the natural frequency. This work will help to learn more about what to do or what is necessary to a real modal analysis and comparation in a more robust CubeSat model.

2 Methodology

Following the rules in the Cal Poly standard (CALYFORNIA POLYTECHNIC STATE UNIVERSITY [4]), was designed a CAD model using the CAD software SOLIDWORKS 2016 and the FEM analysis was made in the software Ansys 18.1. The CubeSat model wasn’t a model to be built yet, in a real model the joints could not fix the plates and the thickness of 1 mm in the plates couldn’t withstand the loads that a satellite should withstand. So, all the analysis made in this work was just to applicate in a future work. Although was made some changes in the design, was tried to respect the principal characteristics of a 2U CubeSat in the Cal Poly standard. The dimension of the structure was a rectangular box of 200mm x 100mm x 100mm (the height should be 227 mm in California Polytechnic State University [4], but was chosen to use 213mm, so there was a height of 200mm without the foot), the dimension of the rail (the feet principally) was of 8.5mm x 8.5mm with a chamfer of 1mm. The material used in the tests was a general Aluminum Alloy from the Ansys engineering data. The Table 1 shows the comparation of the aluminum alloy used and the aluminum alloy 6061-T6 used in Quiroz-Garfias [2] and Chiranjeeve [8]:

Table 1. Ansys Aluminum Alloy and Aluminum Alloy 6061-T6

Mechanical Properties	Ansys Aluminum	6061-T6
Density (g/cm^3)	2.77	2.7
Modulus of Elasticity (GPa)	71	69
Tensile Yield Strength (MPa)	280	270
Poisson's ratio	0.33	0.33

The advantages using the material of Ansys engineering data is that the material has other configurations that may make a more accurate simulation, and the Table 1 shows that the properties aren't so different.

Firstly, was generated the modal frequencies of the structure like Quiroz-Garfias [2], Bürger [6], De Aguiar [7] and Chiranjeeve [8] did. Quiroz-Garfias [2] generated all the natural frequencies of his CubeSat until the closest to 2kHz because his sine sweep test was made until 2kHz. In this work was generated until the 30th mode of vibration, that was almost in 1.5kHz. Due the low capacity of the equipment in the LMV is not possible to test large vibration bands, so in this simulation was just analyzed the mode shapes until the closest to 1 kHz.

This analysis consists of a simple view of the natural frequency, maximum deformation, maximum Von Mises stress and the mode shape in a vibration band of 1kHz. The modal analysis was based in Shepenkov [9] and Boujemai [10], but this approach is very different from them, because their work had different objectives. To compare this result, was also simulate the structure with some loads in the sides of the structure. Since there weren't information about the load of the launch vehicle, was used a load based in Quiroz-Garfias [2], in their work they supposed the worse load in his 3U CubeSat as 294.3N in a vertical arrangement. So, was chosen a load of 196.2N, the equivalent load to a 2U CubeSat, and was applied this load as in vertical arrangement as in a horizontal arrangement, and all as a compressive force. The simulations were performed in the following situations: No loads; Load in a side (any side because of the symmetry); Load in two opposite sides; Load in the base; Load in the top; Load in both base and top.

Then this simulation was done again, this time using a model of circuit plate, so now was possible to see how some mode shapes affected a circuit plate of a satellite. The model was based just in a thin plate that could fit inside the CubeSat. The material used in the simulation was polymer FR-4 (based in Bürger [6]), own of the Ansys engineering data.

In addition to this work was also verified the influence of a slacking in a foot of the CubeSat to the deployer. In this part was done a modal analysis to slacking in a foot, with no load and the circuit plate. But using Harmonic Analysis, also in the Ansys, was done with circuit plate (WP) and with no circuit plate (NP), and with slacking in 1, 2 and 3 feet. Was simulated the displacement of the base rails in the vertical axis and in a frequency band of 1kHz with an interval of 0.5 Hz.

3 Results and discussion

The mass calculated using the volume information in SOLIDWORKS 2016 and the properties of the materials used was of 0.22kg. To find a load of 196,2 N should be necessary a load of 89.7g, a very large load compared to the load that are related in the literature about simulations in payloads. In addition to the fragile structure design was found very large Von Mises stresses.

Comparing the simulations with circuit plate (WP) and with no circuit plate (NP) we observed some characteristics. The addiction of a circuit plate influenced the first mode shapes, but in some modal frequency they became the same again. The structure WP also decreased the deformation and the Von Mises stresses in all simulation loads.

About the influence of the load conditions is shown here only the results to WP structures. The most difference noted was in the slack in a base rail of the satellite, in this condition appears a "extra" mode shape, before getting around the last natural frequency (closer to 1kHz). The results of the load conditions didn't have many differences between their self and with no load. It was also observed that starting from the 7th mode shape, the deformation and stress became bigger. With a specific analysis only in the circuit plate, was observed that until the 16th, in the mode shapes that appear between the frequencies 693 Hz and 702 Hz, the effects of energy of vibration concentrates in the circuit plate causing large deformation and stress. The Fig. 1 shows the deformation intensity in the circuit plate, in the structure WP for all the simulation conditions. The Fig. 2 shows the mode were all the energy concetrate in the circuit plate.

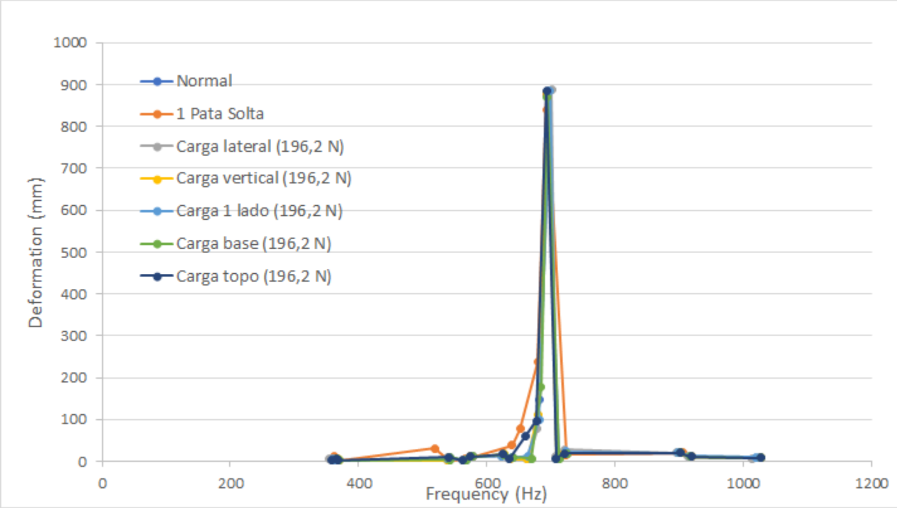

Figure 1. Natural frequency x deformation to simulation condition.

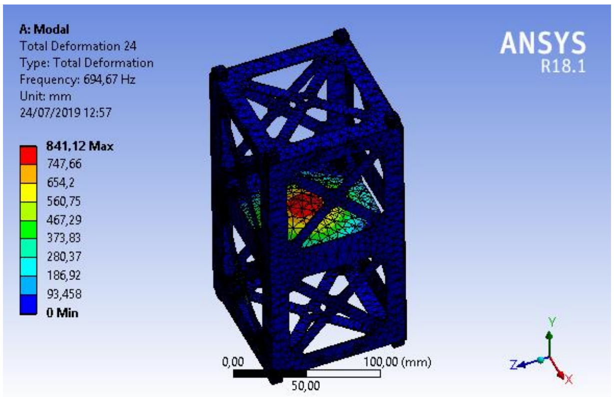

Figure 2. 12th mode of a structure with circuit plate.

In the Harmonic Analysis to slacking feet, was observed that how much more the number of slack base rail increases, lower is the frequency which occurs the biggest displacement in the band of 1kHz. However, wasn't found any correlations to the displacement magnitude to the number of slack foot because it varied in the structure WB and NP.

4 Conclusion

In this work was made analysis in an ordinary CubeSat structure to future studies in a real analysis to a CubeSat that is going to be developed in the LMV. Was used the CAD software SOLIDWORKS to create and the FEM software Ansys to do modal and harmonic analysis.

The results relatives to strength are not very relevant, because of the fragile of structure. The Tensile Yield Strength of the material used is 280 MPa, and in all the simulations the Von Mises stress was far above that, still exceeding the Ultimate Tensile Strength.

It is concluded that the loads haven't much influence in modal analysis, as well his orientation, but the slack in the base rails may make some difference. The presence of a simple circuit plate of FR-4 can change the deformation and stress due the vibration, but this is because the structure without circuit plate have not enough support, so changes in the inside sustentation can improve the structure.

The results also showed mode of vibration that affect directly the circuit plate. These frequencies should be avoided, because the circuit is the most fragile part of the satellite, and damage in this area may compromise the mission.

All the basis to the study was not really planned, but the results can be used in more relevant studies in CubeSats, as well gives a new contribution to the study area. These results will help the LVM in the analysis of the CubeSat that is going to be built. More modal analysis is going to be done, since the structure will be completely modified to a more robust one, and comparations between the modal analysis and the vibration tests (base vibration) can be performed.

Acknowledgements

The authors thank the support and for all the help of the LVM/UFC team.

References

[1] DAVID, L. Cubesats: Tiny Spacecraft, Huge Payoffs. **SPACE.com**. Disponível em: <https://www.space.com/308-cubesats-tiny-spacecraft-huge-payoffs.html>. Acesso em: 12 jul. 2019.

[2] QUIROZ-GARFIAS, C.; SILVA-NAVARRO, G.; RODRÍGUES-CORTÉS, H. Finite Element Analysis and Design of a CubeSat Class Picosatellite Structure. In: International Conference on Electrical and Electronic Engineering, 4, 2007, Mexico City. **Anais**. Mexico City: Institute of Electrical and Electronics Engineers, 2007. p. 294-297.

[3] HANSEN, F. Danish Small Satellite Programm. In: BÜRGER, E. E. *et* al. Development and Analysis of a Brazilian CubeSat Structure. In: International Congress of Mechanical Engineering, 22, 2013, Ribeirão Preto. **Anais**. Ribeirão Preto: Associação Brasileira de Engenharia e Ciências Mecânicas.

[4] CALYFORNIA POLYTECHNIC STATE UNIVERSITY. **CubeSat Design Specification**. California, 2015.

[5] SARAFIN, T. *et al.* **Vibration Testing of Small Satellites**. Littleton, 2017.

[6] BÜRGER, E. E. *et al*. Development and Analysis of a Brazilian CubeSat Structure. In: International Congress of Mechanical Engineering, 22, 2013, Ribeirão Preto. **Anais**. Ribeirão Preto: Associação Brasileira de Engenharia e Ciências Mecânicas.

[7] DE AGUIAR, L. R. C. **Análise Estrutural Estática e Dinâmica de um Satélite de Baixa Órbita**. 2016. Undergraduate Thesis (Bachelor of Aerospace Engineering) – Universidade de Brasília, Brasília, 2016.

[8] CHIRANJEEVE, H. R.; KALAICHELVAN, K.; RAJADURAI, A. Design and Vibration Analysis of a 2U-CubeSat Structure Using AA-6061 for AUNSAT-II. **Journal of Mechanical and Civil Engineering**, Chennai, p. 61-68, 2014.

[9] SHEPENKOV, V. **Vibration Modal Analysis of a Deployable Boom Integrated to a CubeSat**. 2013. Master Thesis (Master of Science and in Engineering Mechanics) – KTH Royal Institute of Technology. Sweden, 2013.

[10] BOUDJEMAI, A. *et al*. Modal analysis and testing of a hexagonal honeycomb plates used for satellite structural design. **Materials and Design**, Arzew, v. 35, p. 266-275, 2012.