A Comparative Experimental - Numerical Analysis on the Vibration Behaviour of a Composite Satellite Subset

SORIN DRAGHICI[1], IOAN PARAUSANU[1], FLORIN BACIU[1], HORIA ALEXANDRU PETRESCU[1], ANTON HADAR[1,2], STEFAN DAN PASTRAMA[1]*

[1]University Politehnica of Bucharest, Department of Strength of Materials,313 Splaiul Independentei, 060032, Bucharest, Romania

[2]Academy of Romanian Scientists, 54 Splaiul Independentei, 050094, Bucharest, Romania

The paper describes a comparative experimental - numerical analysis to study the vibration behavior of a fiberglass/polyester composite box housing electronic components inside a satellite. A finite element model was developed in order to predict the natural modes and the values of the natural frequencies of the structure. In order to perform the numerical analyses, specimens made of the analyzed composite material were tested to obtain the elastic constants. The numerical values of the natural frequencies were further compared to those measured experimentally using a PULSE modal analysis system. The differences between results were less than 9%, validating thus the proposed numerical model, which can be further used to predict the behavior of the subassembly subjected to other loads occurring during the launching phase and on the orbit, as accelerations and/or thermal cycles. Since the order of magnitude of the first natural frequencies is important in understanding and evaluating the performance of subassemblies mounted inside satellites, such a study is necessary in the design phase of these structures.

Keywords: satellite, subassembly, vibration behaviour, composite materials, finite element

In the space industry, there is a constant tendency regarding weight reduction of space shuttles, including microsatellites. This results in the integration of advanced materials such as carbon fiber reinforced polymer (CFRP) in their structure. Integration of advanced composite material structures in space shuttles is not an easy task, as such structures are subject to critical operating conditions in space [1].

The cost of launching an object on a Low Earth Orbit (LEO) satellite has been estimated at about 20000 $/kg [2]. For this reason, weight is a critical parameter to be controlled carefully during the design and planning satellite subsets. In this view, the replacement of traditional metallic materials with lighter composites is of great interest.

Inside the space shuttles, there are multiple protective housings for electronics and equipment. These boxes are traditionally made of aluminum. The development of new composite boxes to protect electronic components spacecraft has been analyzed by ESA / ESTEC, in a program to develop new technologies [3]. With the same weight reduction goal, Kim and Lee [4] proposed a monocoque satellite structure composed of many composite sandwich panels, which consist of two carbon fiber/epoxy composite faces and an aluminum honeycomb core, designed to reduce the structural mass and to improve static and dynamic structural rigidity. Katz et al. [5] investigated the possible use of composite materials in the construction of different parts of a satellite by studying the response to hypervelocity impact by space debris for Kevlar 29/epoxy and Spectra1000/epoxy thin film micro-composites with thickness of about 100 μm. Such materials are used in long-duration spacecraft outer wall shielding to reduce the perforation threat.

During their development, manufacturing, and launch to their final operating position in space, satellites experience different types of mechanical, thermal, and electromagnetic disturbances [6]. Among them, vibrations are extremely important and must be considered in the design phase and monitored during service. That is why several researchers drawn their attention to the experimental measurement and/or numerical calculations of the natural frequencies of satellites or their subassemblies. The vibration characteristics of a satellite were studied by Cho and Rhee [6] using both numerical procedures (Finite Element Analyses) and experimental determinations. Moshrefi-Torbati et al. [7] presented vibration measurements for a satellite boom made of 10 identical bays having equilateral triangular cross sections. A satellite model used for remote sensing was designed, modelled and analyzed by Israr [8]. Oda et al. [9] presented a system that uses a Complementary Metal-Oxide Semiconductor (CMOS) camera to measure the distortion and vibration of the solar array paddle from the Japan Aerospace Exploration Agency's earth observation satellite *GOSAT*. Vibration tests were performed by Paris, on the Italian Space Agency satellite LARES [10]. The three axes slewing maneuver and the vibration suppression of a flexible satellite with a central rigid body and two flexible appendages was described by Azadi et al [11].

This paper is part of a research which addresses the issue of replacing a subassembly hosting electronics (fig. 1) in a low-orbit satellite, traditionally made of aluminum with a structure made of composite material [12-14]. The studied electronic box is one of the subassemblies of the PROBA 2 (PRoject for On Board Autonomy) satellite [15]. PROBA2 is a small satellite (130 kg) which was launched on November 2, 2009, in a sun-synchronous low Earth orbit at an altitude of 725 km. It is the second satellite in the European Space Agency's series of PROBA low-cost satellites, used to validate new spacecraft technologies [16].

The main purpose of this study is to develop a finite element model to predict the vibration behavior of the subassembly made of composite materials dedicated for space conditions. In order to perform the numerical analyses, the elastic constants of the studied material were measured through tensile tests. The values of the natural

* *email: stefan.pastrama@upb.ro*

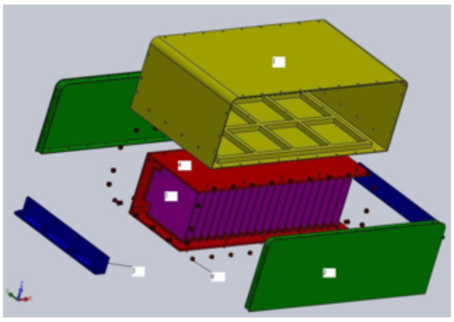
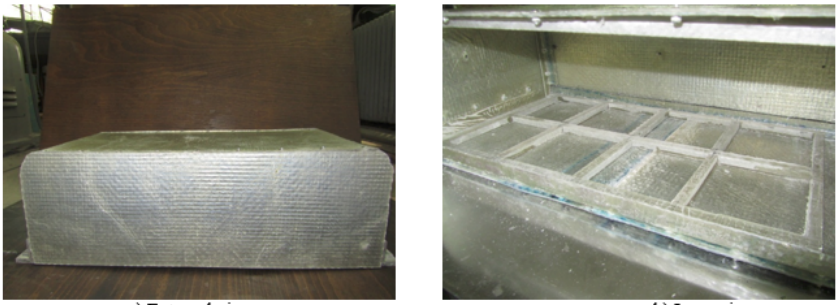


Fig. 1 The subassembly hosting electronics in the satellite



a) External view b) Inner view

Fig. 2. The experimental composite model

frequencies of the structure obtained with finite element analyses were compared to those measured experimentally using a PULSE modal analysis system. Such a study is essential in understanding and evaluating the performance of any engineering product, especially since in this case, a prerequisite for the satellite box is the order of magnitude of the first natural frequencies.

The studied structure

The tested structure was a box made of a multilayered composite material (fig. 2). The following arrangement of layers was adopted on the wall thickness from outside to inside:

- Layer 1: Two laminae of polyester resin reinforced with glass fibers, layer that acts as a heat shield for electronics;
- Layer 2: Aluminum foil which protects the electronic plates from radiation, simulating the behaviour of tungsten layer;
- Layer 3: Three laminae polyester resin reinforced with glass fibers, a composite material that can ensure the structural integrity of the subassembly.

Tensile tests

In order to model the vibration behaviour of the subassembly as accurately as possible, the elastic constants of the materials were experimentally determined through tensile tests. Typical specimens were made both from the composite material and aluminum. Two types of composite specimens (with two and with three layers respectively) were cut both longitudinally and transversally, to obtain the mechanical characteristics on both directions (denoted as 1 and 2). For each type, five identical specimens were manufactured and the obtained results were averaged. To obtain the characteristics on the thickness direction (denoted as 3), another specimen was manufactured from a plate made of 200 layers, and the obtained values were considered both for the two layered and the three layered composite. The following elastic constants were obtained: Young's modulus E, shear modulus G, Poisson's ratioν. Also, measurements of the mass and volume of the samples were undertaken to obtain the mass density ρ.

The specimens are shown in figure 3 and 4. The obtained stress strain curves for each of the five specimens made of composite material and for all studied cases are depicted in figure 5 and 6.

In order to obtain the Young's modulus along the thickness of the composite material, a test piece of 200 layers was made (fig. 7). Due to limitations imposed by the extensometer of the testing machine, in which the specimen could not be mounted, measurements were made using the strain gauge technique. A rosette strain gauge was glued on the specimen and strains were measured using a Spider 8 strain gauge bridge.

Also using a strain gauge rosette, experimental determination of the Poisson's ratio was carried out for a set of five specimens. On each test specimen, subjected to traction in an INSTRON 8801 testing machine, the strains were measured both on the longitudinal direction (direction of the load) and the transversal direction.

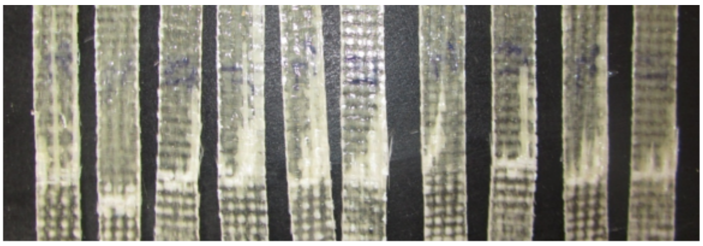


Fig. 3 Two layers composite specimens (longitudinal and transversal cut)

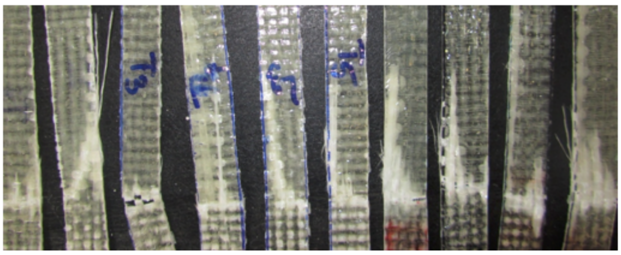
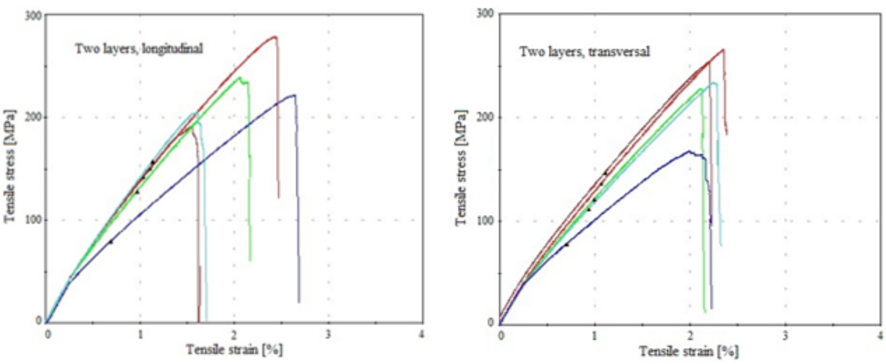
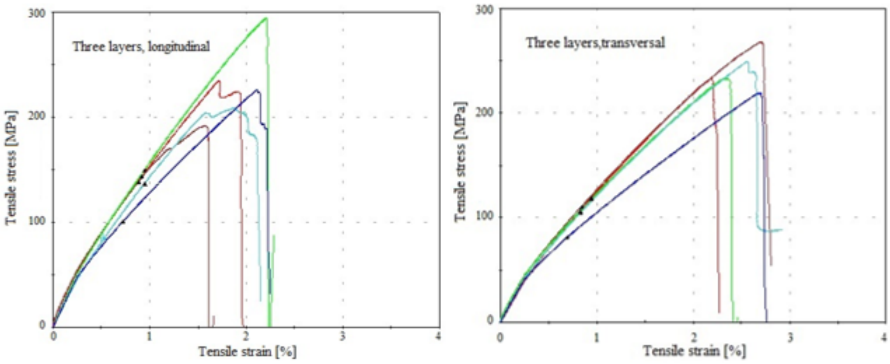


Fig. 4. Three layers composite specimens (longitudinal and transversal cut)



a. b.

Fig. 5 The stress-strain curves for two layered composite material: a. Specimen cut on the longitudinal direction; b. Specimen cut on the transversal direction



a. b.

Fig. 6 The stress-strain curves for three layered composite material: a. Specimen cut on the longitudinal direction; b. Specimen cut on the transversal direction

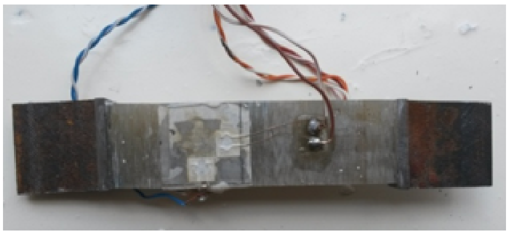


Fig. 7. Specimen used to obtain the Young's modulus along the thickness

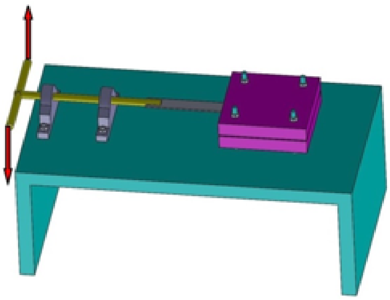


Fig. 8. Experimental set-up for obtaining the shear modulus

For experimental determination of the shear modulus, a special experimental device was designed in order to test a rectangular torsion test specimen. To measure the strains due to the moment of torsion, two strain gauges were glued at 45° with respect to the longitudinal direction of the specimen and a Vishay strain gauge bridge was used. The experimental set-up is illustrated in figure 8.

The obtained elastic constants are listed in table 1, together with the thickness *t* of the sheet for each material.

Modal analysis

For the finite element analysis of the composite structure, a layer based model was chosen for the subassembly and the elastic constants previously obtained were used (fig. 9).

Since for the experimental determinations the subassembly was fixed on the concrete foundation with bolts (simulating the mounting situation inside the satellite), in these places an elastic support was introduced between the model and the platform to which the subassembly is connected. Also, the composite subassembly interior had no electronic plates and outlets.

Because the elements of the box were fixed one to the other by rivets, frictional contact elements were used between these elements. Linear elastic analyses were performed using the finite element code ANSYS [17]. The model, having 389914 nodes and 184937 SOLID 187 elements is shown in figure 10, with the first six natural frequencies. One can notice that the first non-null frequency is the third one.

The elastic natural modes of the structure are detailed in figures 11 -14. For an easier understanding of the

Table 1
THE ELASTIC CONSTANTS OF THE COMPOSITE MATERIAL

	Composite- two layers	Composite- three layers	Aluminum
E_1 [MPa]	16500	19400	68000
E_2 [MPa]	15200	16300	-
E_3 [MPa]	7200	7200	-
G_1 [MPa]	6000	4000	25758
G_2 [MPa]	5000	3500	-
G_3 [MPa]	3150	3150	-
ν_1	0.129	0.129	0.32
ν_2	0.118	0.109	-
ν_3	0.33	0.33	-
ρ [g/cm^3]	1.524	1.884	2.77
t [mm]	1	1.42	0.08

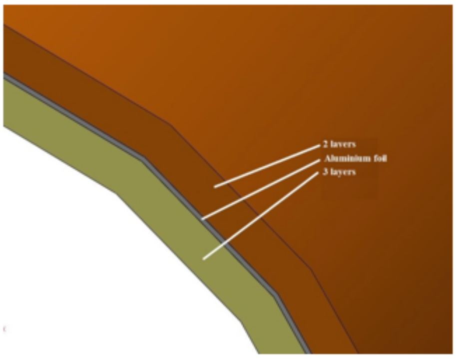


Fig. 9 Layers modeling in the composite material subassembly

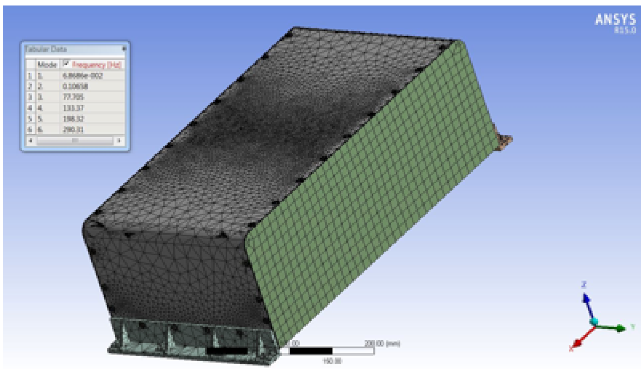


Fig. 10. The first six natural frequencies of the composite structure

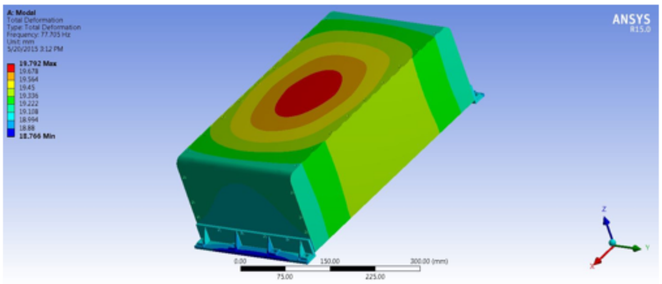


Fig. 11 The first elastic natural mode of the composite structure

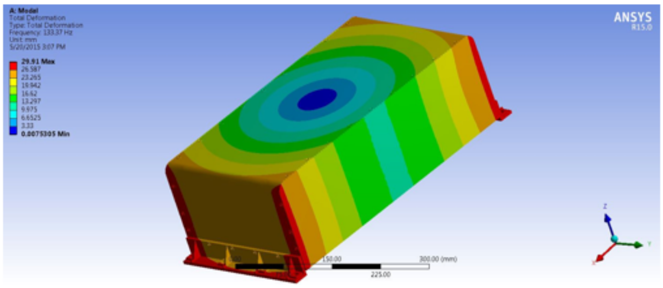


Fig. 12 The second elastic natural mode of the composite structure

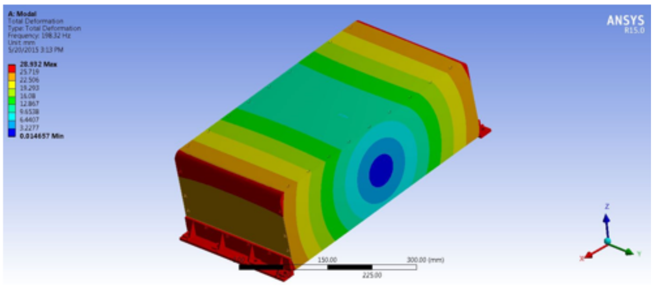


Fig. 13 The third elastic natural mode of the composite structure

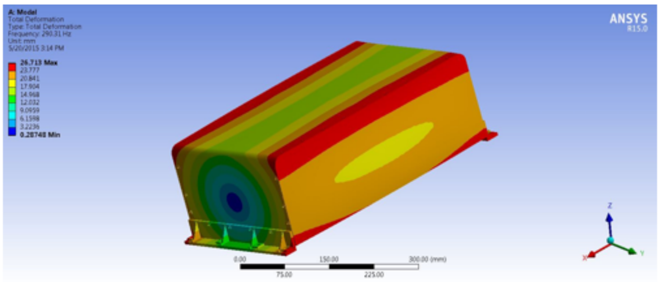


Fig. 14 The fourth elastic natural mode of the composite structure

comments regarding the natural modes, the faces of the box were numbered according to figure 15.

As one can see in figure 11, the first vibration mode different from zero (mode 3) has the same influence on all faces, differences in deformation being very small between the maximum and minimum values. Thus, comparison with the experimental results can be done on any face.

In figure12, one can see that the second non-null mode of vibration affects mostly the edge between faces 2 and 3 (see the maximum displacements in red colour), so the experimental determinations should be made based on excitations on these faces.

The third non-zero mode of vibration shown in figure 13 is emphasized mostly on the edge between faces 1 and 3.

Finally, in figure 14 one can see that the fourth vibration mode affects mostly the edge between face 1 and face 2.

Experimental determinations

In order to validate the proposed numerical model, experimental measurements of the natural frequencies were undertaken for the proposed composite subassembly, which was fixed on a concrete foundation with bolts having a diameter of 8mm, simulating the mounting situation

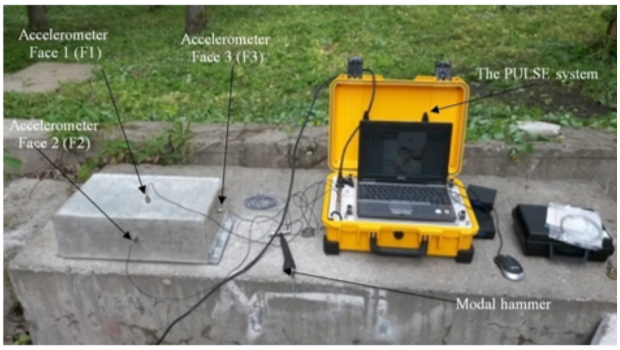


Fig. 15. Experimental measurements of the natural frequencies for the composite subassembly

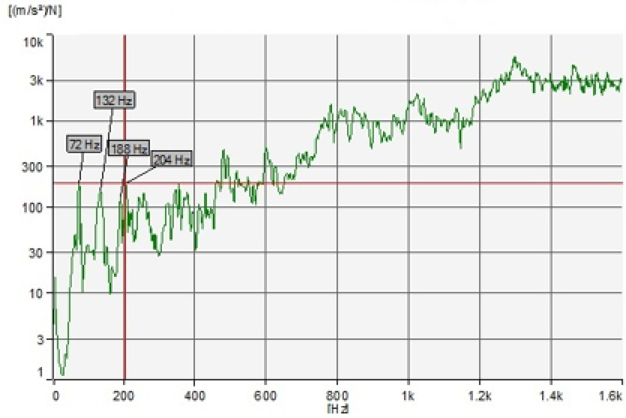


Fig. 16 .The first five natural frequencies experimentally identified on Face 1

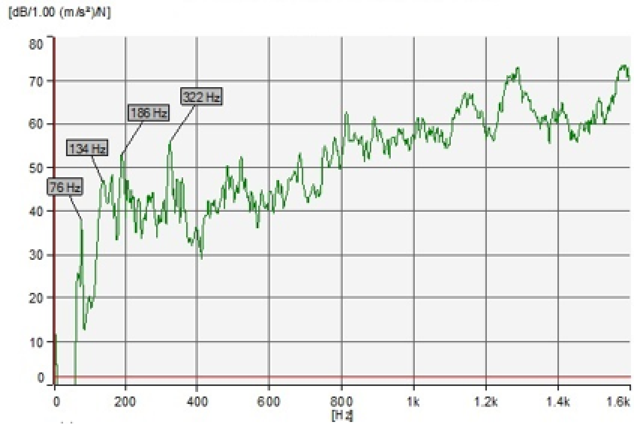


Fig. 17. The first five natural frequencies experimentally identified on Face 2

inside the satellite. The excitation was performed using an 8206 B&K modal hammer with steel head.

For measuring the response, three 4514 B&K piezoelectric accelerometers were mounted on distinct faces of the subassembly, noted F1, F2, F3 (fig. 15). On each face, excitation was done using a modal hammer and the response was measured. The analysis of experimental data was performed using the Brüel & Kjær system PULSE for dynamic analysis and the frequency response curves were plotted.

Although nine frequency response curves were obtained, only those showing the natural frequency on the excited face were shown in figures 16-18.

The values of the experimentally obtained natural frequencies were compared with those that yielded from the numerical analysis previously described. The

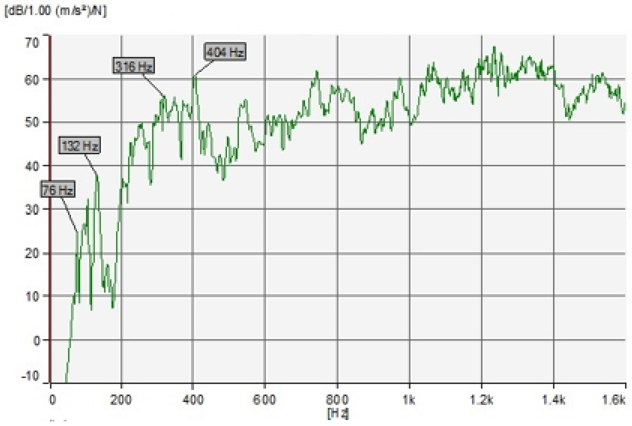


Fig. 18. The first five natural frequencies experimentally identified on Face 3

comparative results, together with the difference between the two sets of results are listed in table 2.

Table 2

COMPARATIVE RESULTS - EXPERIMENTAL vs. NUMERICAL FOR THE COMPOSITE BOX

Mode	Natural frequency [Hz]		Difference [%]
	Experimental	Numerical	
1	76	77.70	2.18
2	132	133.37	1.02
3	188	198.32	5.2
4	316	290.31	8.84

Conclusions

In this paper, the vibration behavior of a fiber glass reinforced composite box housing electronic components inside a satellite was undertaken using both numerical and experimental methods. It was shown that the finite element model conceived for obtaining the natural frequencies of the box can accurately describe the behaviour of the subassembly. By studying the results listed in table 2, one can notice that the difference between the numerical and experimental results is less than 9%, error that is confined to the usual engineering approach. Also, it can be emphasized that the numerical analyses give an indication about the edges or faces of the structure where different modes of vibration have a greater influence. While the first vibration mode different from zero has the same influence on all faces, the other ones affects mostly two faces, as it was shown in the previous paragraphs. This analysis is important for subsequent experimental determinations in order to have an indication on the places where accelerometers should be mounted.

It can be concluded that the model accurately predicts the real vibration behavior of the composite box, and can be further used in assessing the behavior of the subassembly subjected to other loads occurring during the launching phase and on the orbit, as accelerations and/or thermal cycles.

References

1. PIPPIN, G., Space environments and induced damage mechanisms in materials, Prog. Org. Coat. **47**, 2003, p. 424–431.

2.ERIC, J. J., HULL, R.J., ANDERL, T.R., Laser hardened materials evaluation laboratory simulates space environment for advanced materials, space system testing, Materials in Space: How Air Force research is Maintaining America's Advantage in Space, AMPTIAC Quart., **8**, 2004, p. 31-34.

3.BRANDER, T., GANTOIS, K., KATAJISTO, H., WALLIN, M., CFRP electronics housing for a satellite, in Proc. European Conference on Spacecraft Structures, Materials and Mechanical Testing 2005 (ESA SP-581). 10-12 May 2005, Noordwijk, The Netherlands, Karen Fletcher, Ed., Bibliographic Code: 2005ESASP.581E.144B

4.KIM, B. J., LEE, D. G., Development of a satellite structure with the sandwich T-joint Compos. Struct., **92**, 2010, p. 460-468.

5.KATZ, S., GROSSMAN, E., GOUZMAN, I., MURAT, M., WIESEL, E., WAGNER, H.D., Response of composite materials to hypervelocity impact, Int. J. Impact Eng. **35,** 2008, p. 1606–1611.

6.CHO, H.K., RHEE, J., Vibration in a satellite structure with a laminate composite hybrid sandwich panel, Compos. Struct. **93**, 2011, p. 2566–2574.

7.MOSHREFI TORBATI, M., KEANE, A.J., ELLIOTT, S.J., BRENNAN, M.J., ANTHONY, D.K., ROGERS, E., Active vibration control (AVC) of a satellite boom structure using optimally positioned stacked piezoelectric actuators, J. Sound Vib., **292**, 2006, p. 203–220.

8.ISRAR, A., Vibration and Modal Analysis of Low Earth Orbit Satellite, Shock Vib., **2014**, 2014, Article ID 740102,://dx.doi.org/10.1155/2014/740102.

9.ODA, M., HONDA, A., SUZUKI, S., HAGIWARA, Y., Vibration of Satellite Solar Array Paddle Caused by Thermal Shock When a Satellite Goes Through the Eclipse, in Advances in Vibration Engineering and Structural Dynamics, Chapter 13, Francisco Beltran-Carbajal, Ed., ISBN 978-953-51-0845-0, 2012.

10.PARIS, C., Vibration tests on the preloaded LARES satellite and separation system, in Aerosp. Sci. Technol., **42**, 2015, p. 470–476.

11.AZADI, M., FAZELZADEH, S.A., EGHTESAD, M., AZADI, E., Vibration suppression and adaptive robust control of a smart flexible satellite with three axes maneuvering, Acta Astronaut. **69** 2011, p. 307–322.

12.DRAGHICI, S., BACIU, F., VOICU, R., HADAR, A., Modeling the behavior at launching for a satellite's subassembly, in Conference Proceedings of the Academy of Romanian Scientists, Productica, 2014, p. 39-50.

13.DRAGHICI, S., PETRESCU, H.A., VOICU, R., HADAR, A., Behaviour analysis for a subassembly in a LEO satellite, U. Politeh. Buch. Ser. D, **77**, 2015, p. 135-144.

14. DRAGHICI, S., PARAUSANU, I., PASTRAMA, S.D., ZAHARIA, L., HADAR, A., Vibration behavior of a low-orbit satellite subassembly, U. Politeh. Buch. Ser. D, **78**, 2016, p. 115-124.

15.GANTOIS, K., TESTON, F., MONTENBRUCK, O., VUILLEUMIER, P., BRAEMBUSSCHE, P.V.D., MARKGRAF, M., PROBA-2 - Mission and New Technologies Overview: Small Satellite Systems and Services, The 4S Symposium, Chia Laguna, Sardinia, Italy, Sept. 25-29, 2006.

16.***http://www.esa.int/Our_Activities/Space_Engineering_Technology/Proba_Missions/About_Proba-2 .

17.***ANSYS Mechanical ADPL Theory Reference, ANSYS, Inc., Canonsburg, PA 15317, USA, 2013

Manuscript received: 12.04.2016