Modal Analysis of Space-rocket Equipment Components

A.A. Igolkin*, A.I. Safin and A.B. Prokofiev

Heat Engineering and Heat Engines Department, Samara National Research University, Samara, Russia

*Corresponding author: igolkin97@gmail.com

Abstract. In order to prevent vibration damage an analysis of natural frequencies and mode shapes of elements of rocket and space technology should be developed. This paper discusses technique of modal analysis on the example of the carrier platform. Modal analysis was performed by using mathematical modeling and laser vibrometer. Experimental data was clarified by using Test.Lab software. As a result of modal analysis amplitude-frequency response of carrier platform was obtained and the parameters of the elasticity was clarified.

1. Introduction
A significant number of space-rocket equipment (SRE) failures is connected to fatigue failures of parts under the alternating stresses caused by vibroacoustic loads. In order to prevent vibration breakdowns, it is necessary to analyze the natural frequencies and vibration modes of the SRE components. Modal tests are mandatory in most countries while in Russia the requirements for such normative documentation (ND) are still being developed nowadays. In this regard, the issue of creating modal analysis technology is of great importance and interest, since it improves the accuracy and efficiency of vibroacoustic testing and reduces the implementation time.

According to the research of the Experimental Mechanics Community [1], the determination of the dynamic characteristics of spaceships and their components can be divided into two categories: strength analysis and analysis of the satellite structural stiffness. The purpose of strength analysis is to confirm the integrity and performance of the components under vibration loads, comparable to those which appear during the flight. The dynamic analysis of the satellite structural stiffness poses the task of determining its modal parameters (natural frequencies and vibration modes, as well as damping at these frequencies). The purposes of determining modal parameters are the following:

1. Determination of resonance frequencies;
2. Checking the accuracy of the mathematical finite element model and its elements;
3. Determination of the value and change of modal parameters of remote elements having a significant spatial extent [2].

Finite element model and experimental modal verification are used for improving pump vibroacoustic performances in research paper [3]. In paper [4] authors describe in detail the contactless laser vibrometry method as applied to the car surface using the three-component laser scanning vibrometer for verify the vehicle finite-element model. Paper [5] presents research on diagnostic parameters using the Doppler Effect laser vibrometer for monitoring conditions of lightweight aircraft engine blades made of composite materials in flight.

In this paper we consider an example of determining the resonance frequencies and refine the mathematical finite element model. The object of the study is the carrier platform (CP), designed for
fixing the elements of the spaceship apparatus. It should be noted that a laser vibrometer was used in many researches in order to increase the accuracy and speed of modal analysis. The need for using laser vibrometry is determined by the fact that the contact vibration sensors installed on the thin-walled structures of the satellite introduce significant interference due to the additional mass. In addition, the use of a scanning laser vibrometer allows the use of a large number of vibration detection points (and, hence, a more accurate pattern).

The natural frequencies of the component structure are identified by determining and recording the maximum values on the vibration velocity graph from the test report. Modal analysis makes it possible to determine:
- the amplitude-frequency characteristics (by vibration velocity) at points on the surface of the component;
- the amplitude-frequency response averaged over the surface of the component (by vibration speed);
- the vibration form of the component which corresponds to each value of the natural frequency.

2. Methods

Search of the CP for its resonance frequencies is performed using a Polytec PSV-400 laser vibrometer. After the scanning process is completed, the results of vibration measurement of all fragments are connected together. Then transfer functions from the software complex "Polytec Scanning Vibrometer" are imported into "LMS Test.Lab", where calculation of modal parameters is performed. The test procedure for the CP is shown in Figure 1. For the first scheme (Fig. 1a), the CP was suspended on rubber bands without securing to the floor. For the second scheme (Fig. 1b), the measurement object was suspended on an elastic suspension on one side, and its lower part is cantilevered to the base with two hinges on the other side.

A vibration exciter is used as a source of excitation, fixed at the point with maximum displacement when generating vibrations. The vibration exciter in both cases of fixing is set equally to minimize the discrepancy under the experimental conditions.

![Figure 1](image-url)
First, the so-called mode overview, during which the object is scanned in a wide frequency range and with a small resolution in frequency, is performed. At this stage, the frequency bands, in which the resonance of the structure may arise, are determined.

However, this analysis in such a wide frequency range gives a general idea of the characteristics, since it is a preparatory stage in the subsequent refinement of the natural frequencies and vibration modes of the object.

Repeated scanning performed in a narrow frequency range with high resolution in frequency allows us to refine the obtained values of the object natural frequencies. In order to reduce the time of experiment, it is advisable to reduce the number of scanning points.

The final step in the calculation is to determine the shape of the vibrations. To do this, the object is scanned at a previously refined frequency with a larger number of scan points.

A review of the received modes using the Test.Lab (Modal Analysis) software allows to identify stable poles, which are analyzed in accordance with the "modal phase collinearity" (MPC) and "maximum phase deviation" (MPD) indicators. Output data of the program are arrays of obtained values of frequency transfer functions and natural frequency values. Figure 2 shows the frequency response of the structure after processing the data in the program. The solid line shows the data without processing in the program; the dotted one shows the data after processing.

![Figure 2. Amplitude-frequency characteristic of CP after using Test.Lab.](image)

The simulation of the CP natural oscillations is carried out by means of the procedure applied for solving the Lanczos eigenvalue problem which is used in the MSC.Nastran finite element system, which is described in detail in [6]. Search of natural oscillations is carried out in the frequency range from 0 to 500 Hz.

3. Results

As a result of scanning the object from 30 to 500 Hz with the first variant of fixing, the first 3 harmonics of the platform have been identified: 74.4; 113.1; 150.0 Hz. Then the values of the object natural frequencies have been refined and the vibration forms at these frequencies have been determined. The simulation of the experiment in MSC.Nastran have also been performed in the frequency range from 0 to 500 Hz.

Figure 3 shows a comparison of the first vibration form obtained from the experiment and the simulation.

As shown in Figure 3 simulation mode was identical to experimental mode. The same results were found for other modes.
Table 1 compares the first three values of the object natural frequencies obtained from the experiment and the simulation.

Figure 3. First mode during the experiment (top-74.4 Hz) and during the simulation (bottom-81.5 Hz)
As a result of scanning the object in the frequency range from 30 to 500 Hz with the second variant of fixing, the first 3 harmonics of the platform are identified: 38.6; 118.5; 194.8 Hz. Results obtained by using second fixing scheme are summarized in Table 2.

Table 1. Comparison of theoretical and experimental values of natural frequencies

| Mode number | The natural frequency in the experiment (Hz) | The natural frequency in the simulation (Hz) | Δ (%) |
|-------------|---------------------------------------------|---------------------------------------------|-------|
| 1           | 74.4                                        | 81.5                                        | 9.54  |
| 2           | 113.1                                       | 118.8                                       | 5.04  |
| 3           | 150.0                                       | 158.6                                       | 5.73  |

Table 2. Comparison of theoretical and experimental values of natural frequencies

| Mode number | The natural frequency in the experiment (Hz) | The natural frequency in the simulation (Hz) | Δ (%) |
|-------------|---------------------------------------------|---------------------------------------------|-------|
| 1           | 38.6                                        | 36.1                                        | 6.48  |
| 2           | 118.5                                       | 118.5                                       | 0     |
| 3           | 194.8                                       | 191.3                                       | 1.8   |

Figure 4 shows a comparison of the first form of oscillation as a result of experiment and simulation.
Figure 4. First mode during the experiment (top-38.6 Hz) and during the simulation (bottom- 36.1 Hz)

It was found that simulation modes were identical to experimental modes obtained by using second fixing scheme.

4. Conclusions

In this paper we have investigated that:

1) The proposed method of modal analysis of space-rocket equipment components allows determining the resonance frequencies of the object and verifying the accuracy of the mathematical finite element model and its elements.

2) The results obtained are consistent with the result of finite element modeling of the structure in the considered frequency range.

The results indicate the overall correctness of the finite element models. In comparison with experimental data discrepancy of the simulation results does not exceed 10% which indicates the correctness of finite element models.

Acknowledgments

The results of the work were obtained using the grant of the President of the Russian Federation for state support of young Russian scientists (grant number MD-3082.2017.8).

References

[1] Doupe C., Swenson E., George L. and Black J., 2013. Finite element model tuning with 3D mode from FalconSAT-5J. 50thAIAA/ASME/ASCE/AHS/ASC 17th SDM conference proceedings, 17, 7–12.

[2] Testoedov N., Lysenko E.and Berns V., 2016. Diagnostics of structures of space vehicles based on the results of vibration and acoustic tests: monograph. Siberia State Aerospace University: Standartinform.

[3] Safin A., Rodionov L., Makaryants G. and Prokofiev A., 2016. Gear pump modal analysis. Proceedings of ICSV 2016 - 23rd International Congress on Sound and Vibration: From Ancient to Modern Acoustics.

[4] Makaryants G., Safin A. and Shakhmatov E., 2016. Vehicle modal analysis. Proceedings of 22nd International Congress on Sound and Vibration, ICSV 2015.

[5] Makaryants G., Molchanov D. and Safin A., 2016. Journal of the International Measurement Confederation, 95, 383-388.

[6] Peresypkin K. and Kozhichenkova M., 2013. Formation of the shape of the gain of a size-stable platform by means of numerical optimization. Journal of Samara State Aerospace University, 4 (42), 223-229.