The validation of an aerospace structure through the sine vibration analysis

Cristina-Diana BRATU*[,1], Ionut-Cosmin ONCESCU[1], Ion DIMA[1], Alexandra Raluca PETRE[1]

*Corresponding author
[1]INCAS – National Institute for Aerospace Research “Elie Carafoli”, B-dul Iuliu Maniu 220, Bucharest 061126, Romania,
liliceanu.cristina@incas.ro*, oncescu.ionut@incas.ro, dima.ion@incas.ro, petre.alexandra@incas.ro

DOI: 10.13111/2066-8201.2018.10.2.14

Received: 24 April 2018/ Accepted: 15 May 2018/ Published: June 2018

Abstract: *Sinusoidal vibrations represent an ideal case. Technically, it is quite hard to generate pure sinusoidal vibrations without containing other spectral components, called harmonics. Sinusoidal vibrations can appear on propeller, propulsion and turbofan aircraft as well as on helicopters and aerospace structures. They can occur during different phases of flight (take-off, ascent, cruise, landing, etc.). The aim of this article is to present how a structure can be validated by using the mathematical formulas or a FEM software such as PATRAN-NASTRAN (and the equations behind it). As an application, an aerospace structure such as thruster brackets will be analyzed. A sinusoidal signal of 1g was applied on the attachment points and the response was read from the center of gravity of the thrusters.*

Key Words: *Sinusoidal vibrations, frequency response analysis, periodical movement, direct method, SOL 108*

1. INTRODUCTION

Sinusoidal vibrations can be generated by propeller, jet propulsion and turbofan airplanes and also by helicopters.

In the case of *propeller airplanes*, the vibrations measured have a spectrum which is made up of a wideband noise and of several sinusoidal or narrowband lines. Periodic aerodynamic pressure fields are created on the structure of the plane by the flow of air that exists between the blades of the propeller, producing in the same time the peaks of the signals. The amplitude of this flux depends on the point at which the measurement is taken and also on the stage of the flight (take-off, ascent, cruise, landing, etc.).

For the *jet propulsion airplanes*, the strongest vibrations occur during the take-off and ascent, along the vertical axis. The weakest vibrations occur along the horizontal axis of the aircraft. The typical frequency has a value ranging between 60 and 90 Hz and a root mean square of about 5 m/s^2. The value of the frequency depends on the type of airplane [1].

During the cruising phase of the plane, the amplitudes of the vibrations are much lower than for the take-off and ascent phases. However, along the vertical axis, the amplitudes remain stronger. The value of the frequency is also between 60 and 90 Hz.

In the case of *turbofan aircrafts*, it can be observed a continuous rise of the amplitude between 20 and 1000 Hz, and after a decrease. Once again, the weakest vibrations occur along the longitudinal axis and the strongest vibrations along the vertical axis. The vibration signal tends to be made up of a sine wave which is superimposed onto a wideband Gaussian noise [1]. This type of vibration is found on the fighters and is produced by many sources, such as: aerodynamic flow, the engine's noise which is then transmitted by the airplane's bodywork and dynamic responses due to operations (airbrakes, missile launches, etc.). The vibrations determined by *helicopters* are composed of a random wideband noise and sinusoidal lines which are made by the engine, main rotor and tail rotor of the helicopter. The rotation speed of all components remains relatively constant (variation of about 5%) and so does the frequency of the sinusoidal lines that doesn't change much. The fundamental frequency of the sinusoidal lines corresponds to their harmonic frequencies and to the rotation speed of the rotor.

Sinusoidal vibrations represent the simplest form of a periodical movement. This type of vibration has the expression [7]:

$$x(t) = A\sin(\omega t + \varphi) \tag{1}$$

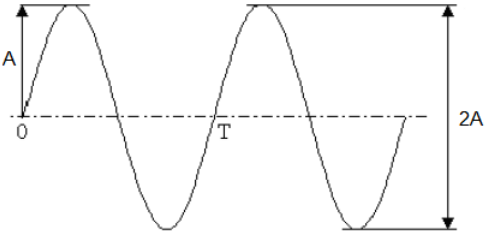

Figure 1. The simplest representation of a sinusoidal vibration [1]

In which:

$$\omega = \frac{2\pi}{T} = 2\pi f\ ; f = \frac{1}{T} \tag{2}$$

And: ω – pulsation (rad/sec),

A – amplitude of the movement (meters),

t – instantaneous value of time (seconds),

φ – initial phase (rad),

T – period (the time of a complete oscillation, seconds),

f – frequency (Hz or s^{-1}).

Generally, $x(t)$ represents the acceleration, but it can be also a velocity, a force or a displacement.

Displacement refers to the variation in position or in distance of an object from a reference axis or a particular point. The displacement range of values, which are between zero (for the zero displacement of a resting system) and the maximum displacement value (zero – peak displacement) can be indicated by the amplitude of the displacement. The interval that exists between the minimum and the maximum values (peak to peak displacement) can also be indicated by the amplitude of the displacement. Velocity is the first derivative of displacement and refers to the variation in displacement over time. Acceleration is equal to the first derivative of velocity or to the second derivative of displacement and refers to the variation in velocity over time. By integration or by differentiation, the two variables are derived from each other as follows:

$$\dot{x}(t) = \frac{dx}{dt} = A\omega \cos \omega t = \dot{A} \cos \omega t = \dot{A} \sin\left(\omega t + \frac{\pi}{2}\right) \tag{3}$$

$$\ddot{x}(t) = \frac{d^2x}{dt^2} = -A\omega^2 \sin \omega t = -\ddot{A} \sin \omega t = \ddot{A} \sin(\omega t + \pi) \tag{4}$$

In the equation (1) the amplitude and the angular speed are constant over time. If the amplitude varies with time, $A=A(t)$, the vibrations can be called amplitude-modulated vibrations and if the frequency varies with time, $f=f(t)$ and $\omega=\omega(t)$, the vibrations can be named frequency-modulated vibrations [2].

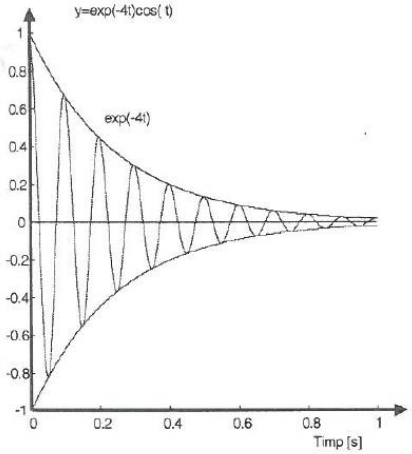

Figure 2. Vibrations modulated in amplitude [2]

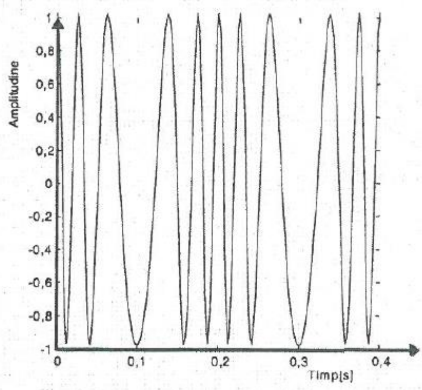

Figure 3. Vibrations modulated in frequency [2]

From these expressions, (3) and (4), we can conclude that acceleration, velocity and displacement are all sinusoidal of period T. There is a phase angle of $\pi/2$ between the velocity and the displacement and there is also a phase angle of π, between the acceleration and the displacement.

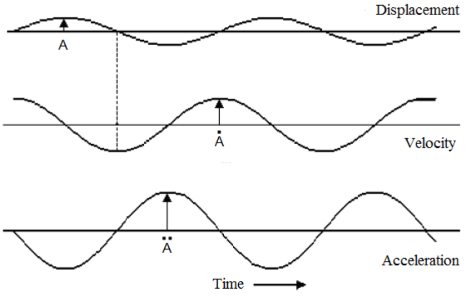

Figure 4. Difference of phase between sinusoidal displacement, velocity and acceleration [1]

From the figure above we can draw the next conclusion: when the displacement has reached its maximum value, the velocity is zero. Also, when the acceleration reaches its maximum level, the velocity is zero. The acceleration is directly proportional with the square of the frequency. The displacement varies as the inverse of the square of the frequency when the acceleration is constant. Therefore, as the frequency increases, the displacement decreases rapidly. Inversely, when the frequency decreases, the displacement increases rapidly.

2. THE METHODOLOGY OF CALCULATING THE DYNAMIC FREQUENCY RESPONSE USING THE FINITE ELEMENT PROGRAM PATRAN-NASTRAN

In the case of the frequency response analysis two different numerical methods can be used: the direct method (SOL 108) which solves the coupled equations of motion in terms of discrete excitation frequency and the modal method (SOL 111) which utilizes the mode shapes of the structure to reduce and uncouple the equations of motion. When using the modal method, the solution for a particular discrete excitation frequency is obtained through the summation of the individual modal responses. To find the structural response for the sinusoidal vibration, the direct method will be used. (SOL 108) [4], [6].

Direct method

Using this method, the structural response will be determined at discrete excitation frequencies by solving a set of coupled matrix equations. The damped forced vibration equation of motion with harmonic excitation, which represents the fundamental equation of the direct method for the dynamic analysis will be considered:

$$[\boldsymbol{M}]\{\ddot{\boldsymbol{x}}(t)\}+[\boldsymbol{B}]\{\dot{\boldsymbol{x}}(t)\}+[\boldsymbol{K}]\{\boldsymbol{x}(t)\}=\{\boldsymbol{P}(\omega)\}e^{i\omega t} \quad (5)$$

From a mathematical point of view, it is more convenient to introduce the load from equation (5) as a complex vector. From a physical point of view, the load can be real, imaginary, or both. The same aspects are valid for the response quantities, too.

In the case of a harmonic motion the solution, has the following form:

$$\{\boldsymbol{x}\}=\{\boldsymbol{u}(\omega)\}e^{i\omega t} \quad (6)$$

where $\{\boldsymbol{x}\}$ is a complex displacement vector. Taking the first and second derivatives of equation (6) the following is obtained:

$$\{\dot{\boldsymbol{x}}\}=i\omega\{\boldsymbol{u}(\omega)\}e^{i\omega t} \quad (7)$$

$$\{\ddot{\boldsymbol{x}}\}=-\omega^2\{\boldsymbol{u}(\omega)\}e^{i\omega t} \quad (8)$$

If the terms of equation (5) are substituted with the expressions from above, the equation below is obtained:

$$-\omega^2[\boldsymbol{M}]\{\boldsymbol{u}(\omega)\}e^{i\omega t}+i\omega[\boldsymbol{B}]\{\boldsymbol{u}(\omega)\}e^{i\omega t}+[\boldsymbol{K}]\{\boldsymbol{u}(\omega)\}e^{i\omega t}=\{\boldsymbol{P}(\omega)\}e^{i\omega t} \quad (9)$$

The equation (9) simplifies after dividing by $e^{i\omega t}$:

$$[-\omega^2\boldsymbol{M}+i\omega\boldsymbol{B}+\boldsymbol{K}]\{\boldsymbol{u}(\omega)\}=\{\boldsymbol{P}(\omega)\} \quad (10)$$

If damping is included or if the applied loads have phase angles the last expression represents a system of equations with complex coefficients. The left expression gives the natural frequencies and the normal modes. The most used method for solving the equation of motion (5) is the complex admittance method [2].

Damping in Direct Frequency Response

The energy dissipation characteristics of a structure are simulated by means of damping. In the case of frequency response, PARAM, G (which defines the overall structural damping) and GE (which defines the element structural damping) on the MAT entry do not form a damping matrix. Instead, they form the following complex stiffness matrix [4], [6]:

$$[\boldsymbol{K}]=(1+iG)[\boldsymbol{K}]+i\sum G_E[\boldsymbol{K_E}] \quad (11)$$

where:

$[\boldsymbol{K}]$ = global stiffness matrix,

G = overall structural damping coefficient (PARAM, G),

$[\boldsymbol{K_E}]$ = element stiffness matrix,

G_E = element structural damping coefficient (GE on the MATi entry).

The overall structural damping coefficient (PARAM,G) will be used for an isotropic structure and the element structural damping coefficient (G_E) will be used for a hybrid structure made of different materials (on the MATi entry). For complex structures, with different materials, it's better to define the overall structural damping coefficient, G, for each natural frequency, from the interested area, using a testing laboratory (SDAMPING, TABDMP1 defined in Patran). An important remark is that the two coefficients G and G_E cannot be used in the same time. The parameters and/or coefficients from above are automatically incorporated into the stiffness matrix and therefore into the equation of motion for the solution when they are specified. In the same analysis all forms of damping can be used but their effects are added together.

3. VALIDATION OF A SPACE STRUCTURE USING A DYNAMIC FREQUENCY RESPONSE ANALYSIS

For this application a structure used to attach the thrusters on the aerospatial structures will be analyzed, see Figure 5. On the interface of bracket with the spacecraft, the FEM was simply supported at the bottom edges as shown in the figure below, with SPC 1, 2, and 3 (translations were blocked). For the dynamic analysis, the nodes on the washer around the spacecraft attachment holes were also blocked in translation with SPC 1, 2 and 3, see Figure 5. The units of measurement used for this model are: the system [Kg, m, seconds], kg/m^3 for density, m/s for speed, m/s^2 for acceleration and PARAM, WTMASS,1. The load represents an acceleration of 1g on all the three global directions (X, Y, Z), with a frequency ranging from 0 to 120 Hz. The analysis uses a critical damping ratio of $\zeta = 0.03$. A sine analysis will be performed using SOL 108 (direct frequency response). Previously, a normal modes analysis will be generated in order to determine the natural frequencies. After performing this analysis, the following quantities can be obtained: accelerations, speeds, displacements, forces and stresses between 0-120 Hz. In this case the results of interest are the accelerations. The resulted accelerations must not exceed 40g, because this value represents the critical value for the proper functioning of the thrusters.

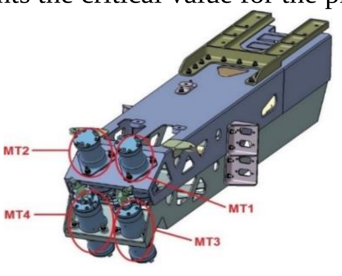

Figure 5. The bracket used at the attachment of thrusters to the space structures

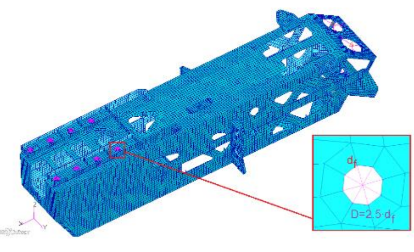

Figure 6. The boundary conditions for the dynamic analysis

The dynamic frequency response analysis will be described in the following steps. After the finite element model of the structure is finished, the proprieties of the material used and the boundary conditions can be introduced. Next, the load case (LC_freq) can be created (which is time dependent, see Figure 7).

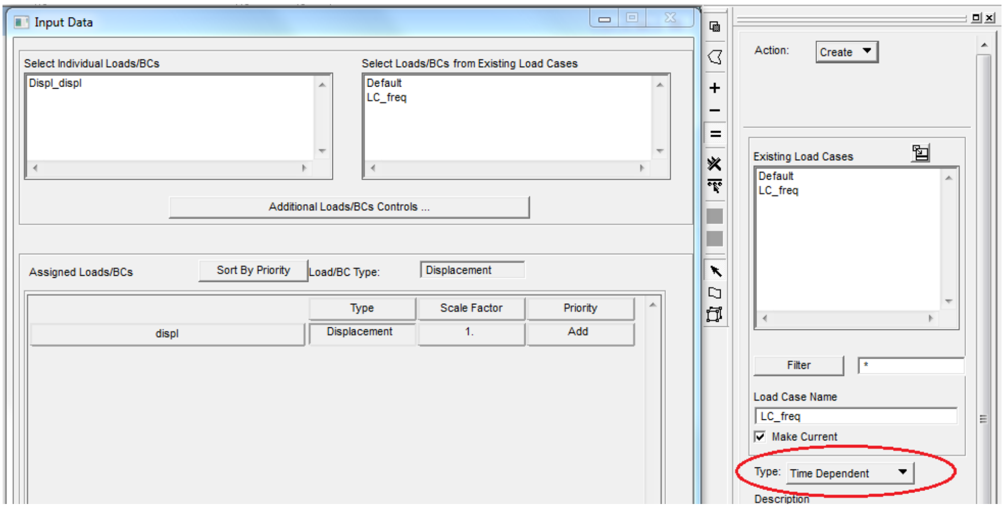

Figure 7. Creating the load case

A frequency dependent field, which helps at the definition of the load, will be created. In the Non Spatial Scalar Table Data, the values describing the time dependence of the load will be introduced.

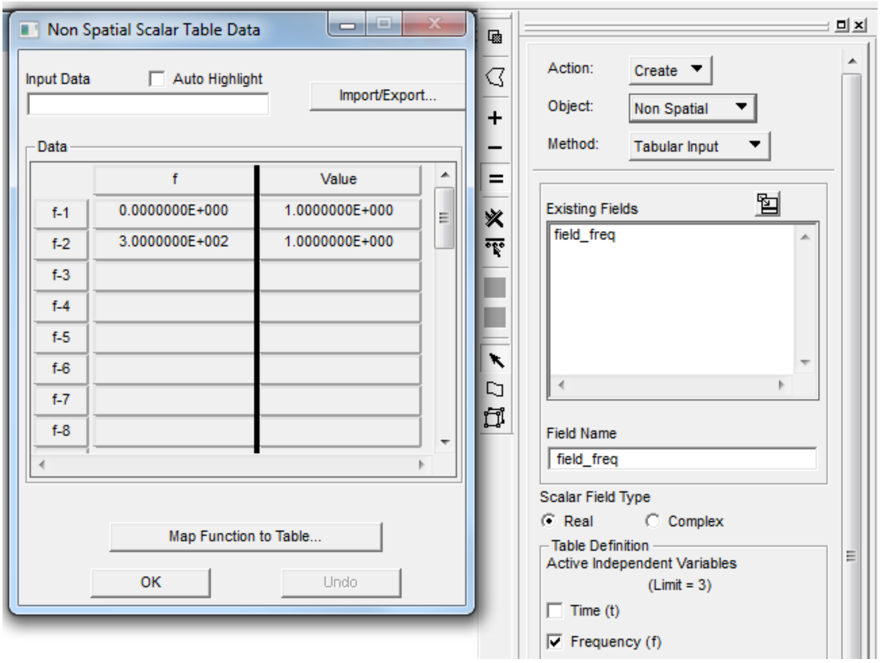

Figure 8. Creating the frequency dependent field

The next step is to define the load, which in this case represents an acceleration of 1g on the Z axis (for the other two axis, X and Y, another two dynamic frequency response analysis will be made).

In the Current Load Case section the load case defined previously will be selected, LC_freq, and in the Time/Freq. Dependence section it will be selected the field defined above, field_freq, from the Time/Freq. Dependent Fields box.

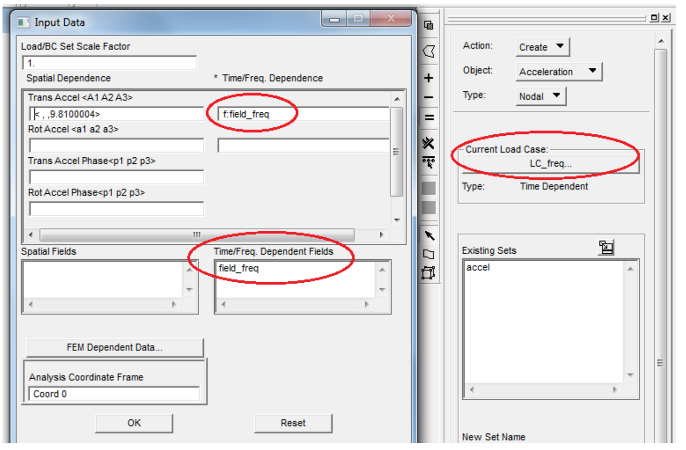

Figure 9. Inserting the load

As an application zone, in the Select Application Region section, it will be chosen the same nodes used to define the boundary conditions, see Figure 10.

To generate an input file for the analysis the following selections are made (see Figure 11): Analysis, Analyze, Entire Model and Analysis Deck.

The type of the solution is defined with the help of Solution Type option. For the dynamic frequency response analysis the Frequency Response option is selected and as a method, the Direct formulation is elected (SOL 108).

The Solution Parameters option is used to pick the adequate parameters, according to the user wishes. For isotropic aluminum alloy, the usual structural damping coefficient is 3% (Struct. Damping Coeff. =0.03) [3], [5].

In the following steps, the load case used in this analysis will be selected and the result cases will be created (Figure 12).

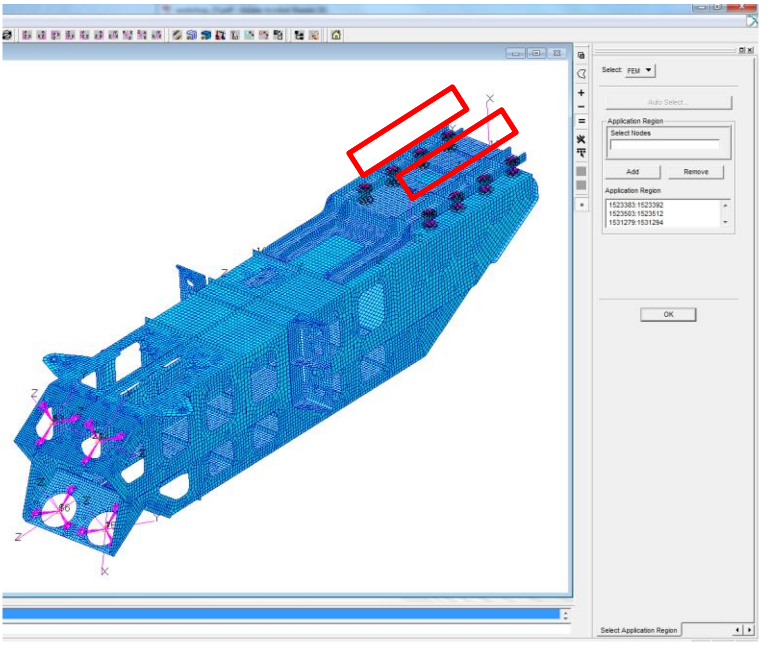

Figure 10. The nodes where the load is applied

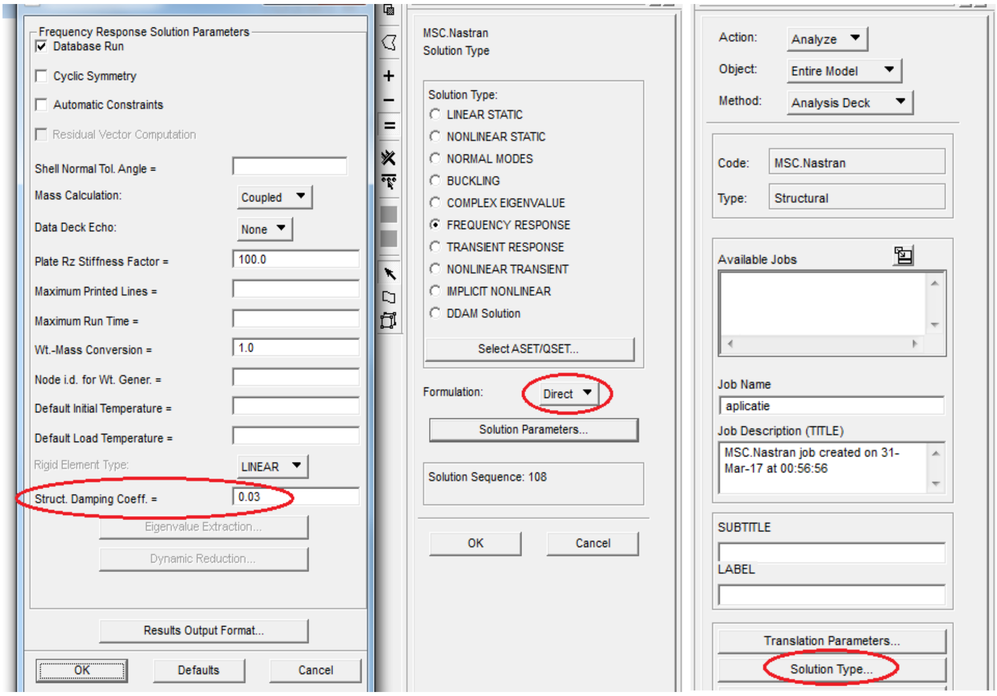

Figure 11. The type of the analysis

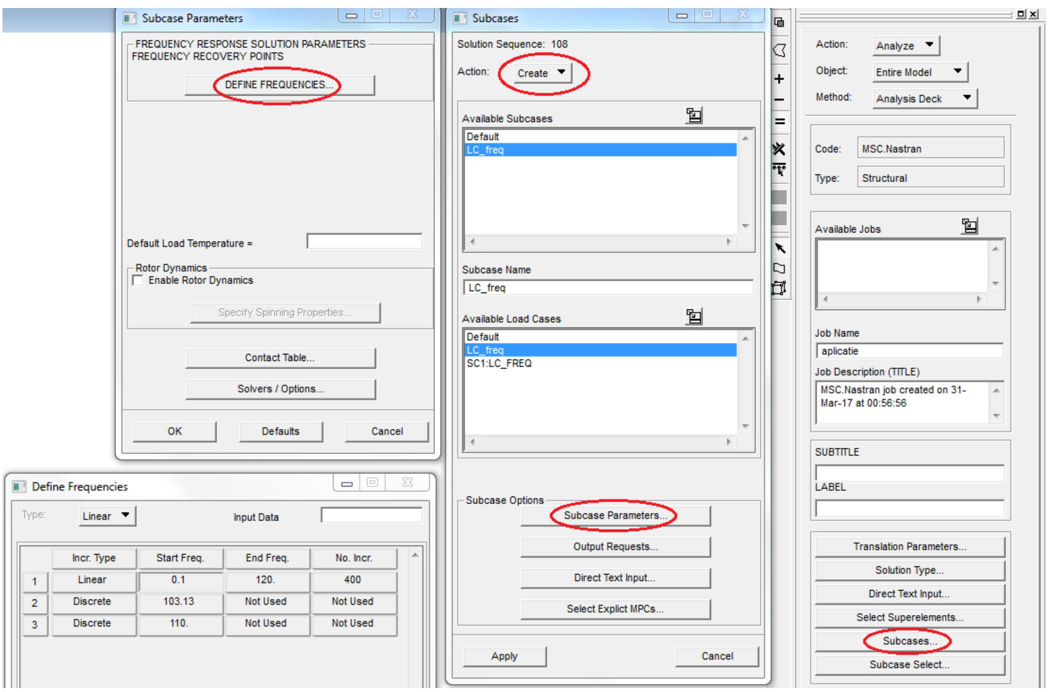

Figure 12. The selection of the load case used in the frequency response analysis

The Subcase option helps in generating the result cases. The frequencies at which the desired information (accelerations, displacements, etc.) can be obtained, are introduced by Subcase Parameters, Define Frequencies.

The inputs are inserted in a table, the same as the one presented in Figure 12. The Linear Increment is used to introduce the desired range of frequencies (0-120 Hz) and the Discrete Increment is used to insert the resonance frequencies calculated in the Modal Analysis.

After generating the Nastran analysis deck file (bdf), it is necessary to modify the output request for the accelerations from SORT1, REAL to SORT2, PHASE. This is needed so that the results are given in magnitude/phase, see Figure 13.

```
SUBCASE 1
   SUBTITLE=analiza_rasp_frecventa_2
   FREQUENCY = 1
   SPC = 2
   DLOAD = 2
   DISPLACEMENT(PLOT,SORT1,REAL)=ALL
   ACCELERATION(SORT2,phase)=1
   SPCFORCES(PLOT,SORT1,REAL)=ALL
OUTPUT(XYOUT)
```

Figure 13. Output request bdf file

The dynamic frequency response analysis is submitted to Nastran. The results are imported to Patran after completion for post-processing.

Next, the graphs used to read the results are going to be plotted (Results section, Figure 14) to read all the acceleration, on the specified axis, corresponding to the frequencies from the range defined above.

All the result cases will be selected. In the Target Entities section, the nodes from which the results will be extracted are the ones from the attachment zone of the thrusters to the bracket (Figure 15).

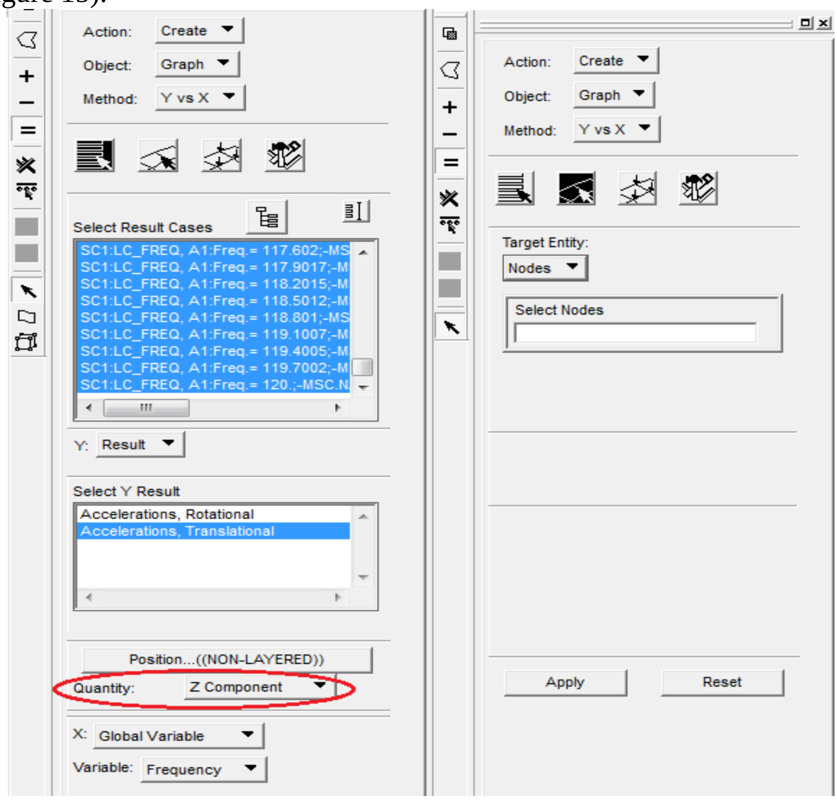

Figure 14. Creating the graph

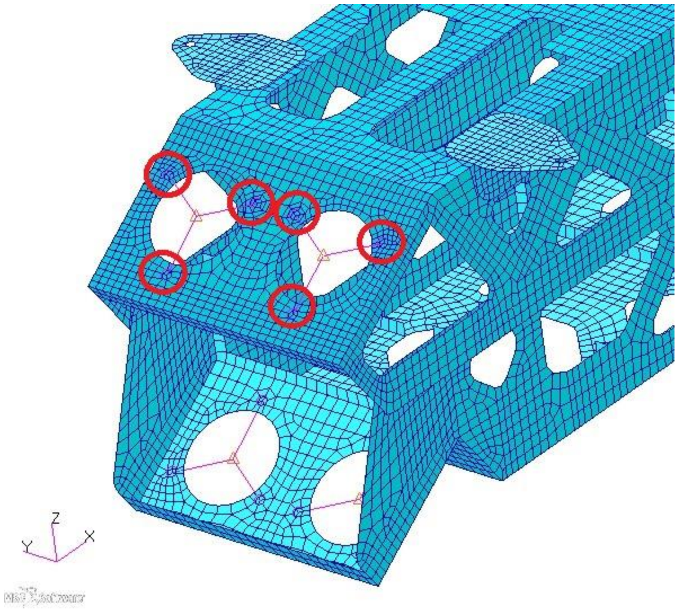

Figure 15. The nodes where the results are extracted

The structure presented is fitted with four thrusters. For every thruster there are three nodes selected from the edges of the thrusters attachment holes, in which the results are going to be displayed.

The Display Attributes and Plot Options (Figure 16) have multiple options of making a graph, dependent on the user’s demands.

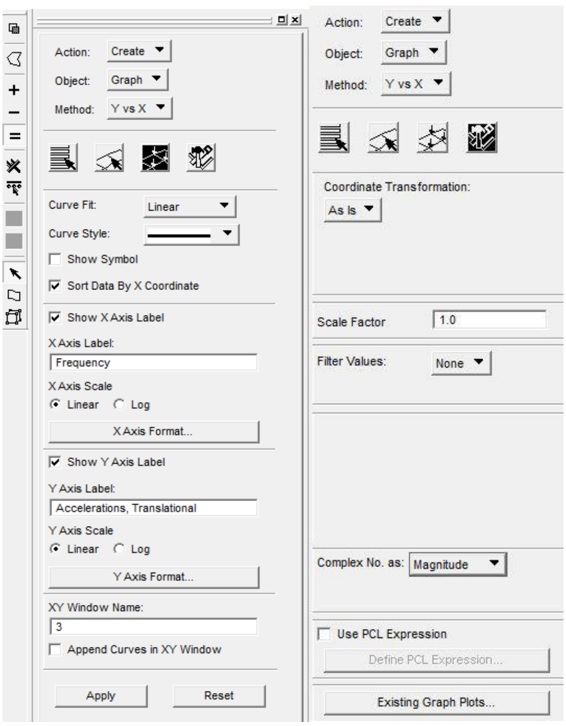

Figure 16. Creating the graph

4. CONCLUSIONS

From the modal analysis, the first frequency mode is obtained at 99.46 Hz, see Figure 17. The modal shape is a bending on the Y axis with a displacement in the Z direction. Because of that, high accelerations are expected when the sinusoidal loading is applied in the Z direction, see Figure 18.

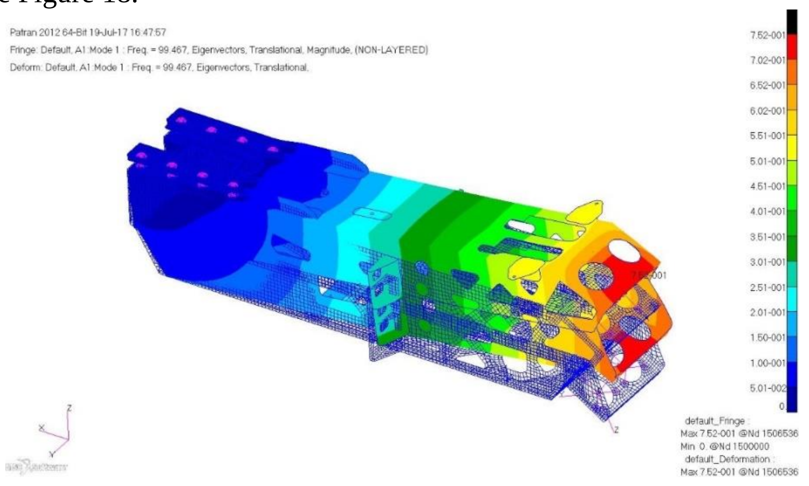

Figure 17. First frequency mode (99.46 Hz)

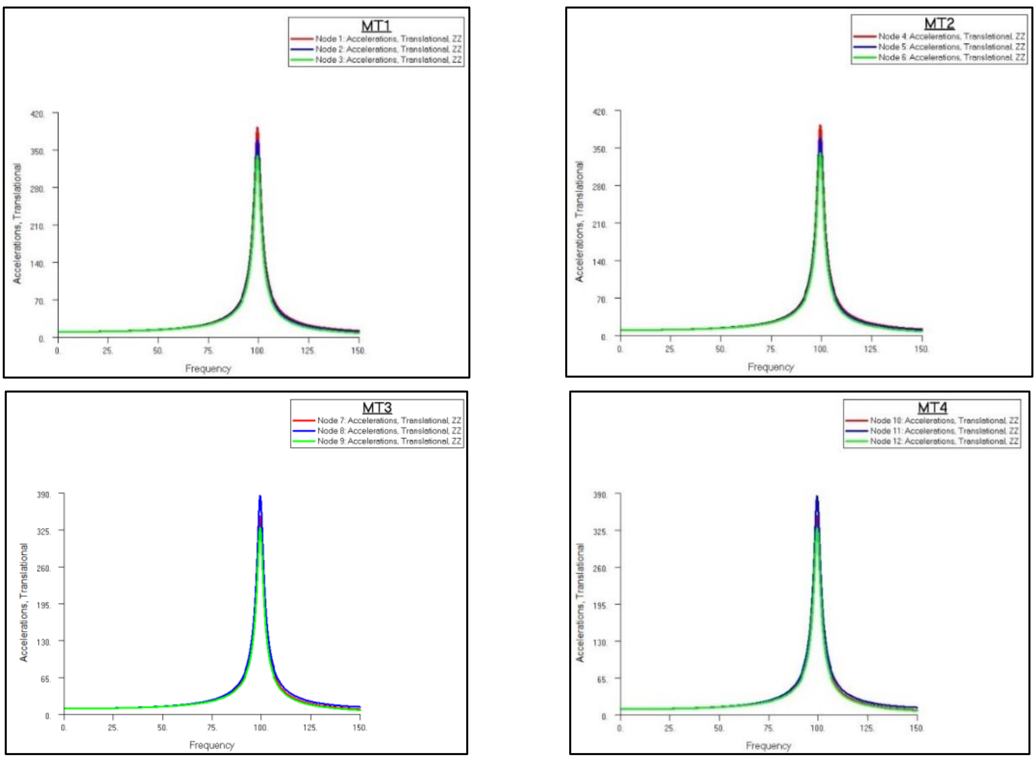

Figure 18. Acceleration response for all four thrusters in the Z direction

The second frequency mode is at 132.2 Hz. The modal shape is a bending along the Z axis, with maximum displacement in the Y direction, combined with torsion, see Figure 19.

The value of the second frequency is above 120 Hz, which is the upper limit of the defined frequency range. Because of that the maximum acceleration response will be achieved above this limit. For this reason, the critical frequency is for the first mode (99.46 Hz).

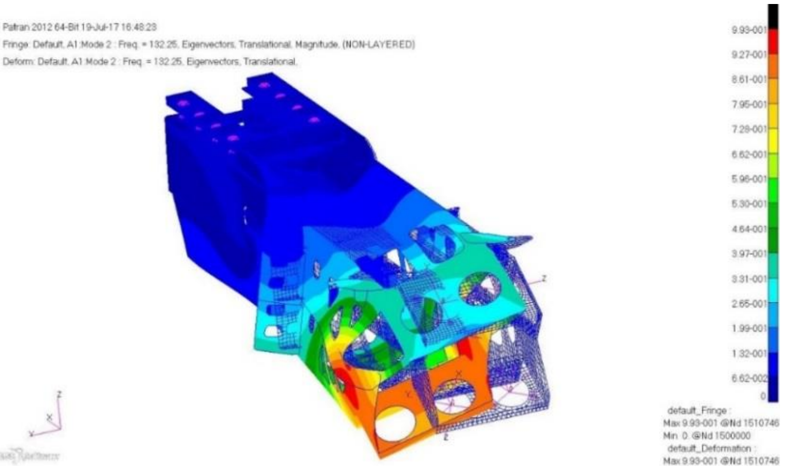

Figure 19. Second frequency mode (132.2 Hz)

The frequency response value for translational acceleration at bracket to thruster interface are summarized in the table below. The values are obtained in the global coordinate system. The maximum values are obtained in the Z direction, as expected. The acceleration values from X, Y direction are taken at 120 Hz because the maximum response is above the defined range, 0 – 120 Hz.

Table 1. Acceleration response for X, Y, Z loading at thruster interface

Thruster no.	**Average acceleration X direction**		**Average acceleration Y direction**		**Average acceleration Z direction**	
	[m/s^2]	**[g]**	**[m/s^2]**	**[g]**	**[m/s^2]**	**[g]**
MT1	11.9	1	35.8	4	365.7	37
MT2	11.9	1	35.8	4	366.5	37
MT3	19.3	2	63.7	6	354.3	36
MT4	19.3	2	63.7	6	354.9	36

This paper presents in depth a procedure for the Sine Vibration Analysis requested for space structures validation. An example of a satellite thruster bracket was analyzed with this procedure. The conclusion is that the bracket is compliant with the requirements of stiffness and strength.

REFERENCES

[1] C. Lalanne, *Sinusoidal Vibration*, vol. **1**, London: ISTE Ltd., 2009.

[2] I. Magheți and M. Savu, *Teoria și practica vibrațiilor mecanice* (*Theory and practice of mechanic vibrations*), Bucharest: Didactical and pedagogical publishing, 2007.

[3] * * * ESA-ESTEC, Space engineering - Spacecraft mechanical loads analysis handbook, European Cooperation for Space Standardization, Noordwijk, The Netherlands, 2013.

[4] * * * MSC Nastran Dynamic Analysis User's Guide, MSC Software, 2014.

[5] P. S. Thomas, *Spacecraft structures and mechanisms*, Microcosm Press, 2011.

[6] * * * MSC Nastran Quick Reference Guide, MSC Software, 2016.

[7] R. Voinea, D. Voiculescu and V. Ceausu, *Mecanica* (*Mechanics*), Bucharest: Didactical and pedagogical publishing, 1975.