Project 3

Dynamic analysis of vibrating structure

Introduction

The aim of this project is to perform dynamic analysis of tethered satellite structure. The structure represents the support of an instrument which is represented by a mass mounted on a satellite. It consists of an arm with the mass at the right end, and it is clamped at the left end. A stiffening bar is hinged to the structure of the satellite at the left end and hinged to the beam at the right end. The material is aluminum alloy (2024 T6) and sections are:

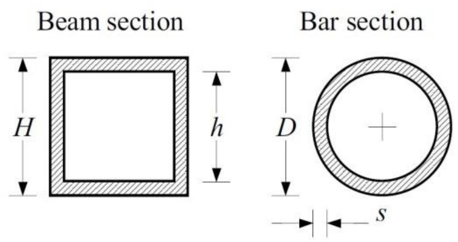

Figure 1: Sections

The dynamics analysis is performed separately in plane X-Z and X-Y. In the X-Z plane the axial and flexural behaviors are investigated whereas in X-Y plane only the flexural one is studied because the reaction forces of the bar acts only in X-Z plane.

Modelling

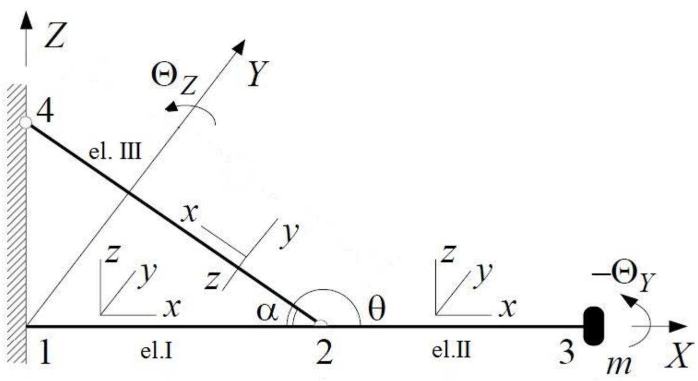

Figura 2: Structure

The structure is modelled with two beam elements (I and II) and one bar element (III). In order to have consistency between local and global reference frames the rotational DOFs of each element is considered positive in X-Y plane and negative in X-Z plane, so it is possible to have the same stiffness matrices of beam elements. Moreover, the local stiffness matrix of the bar has to be rotated in order to have the same orientation between the local and global reference frames. The assembling procedure of global matrices is performed both in the X-Z and X-Y planes using a MATLAB script that works with maps shown below that link the local element DOFs to the global ones.

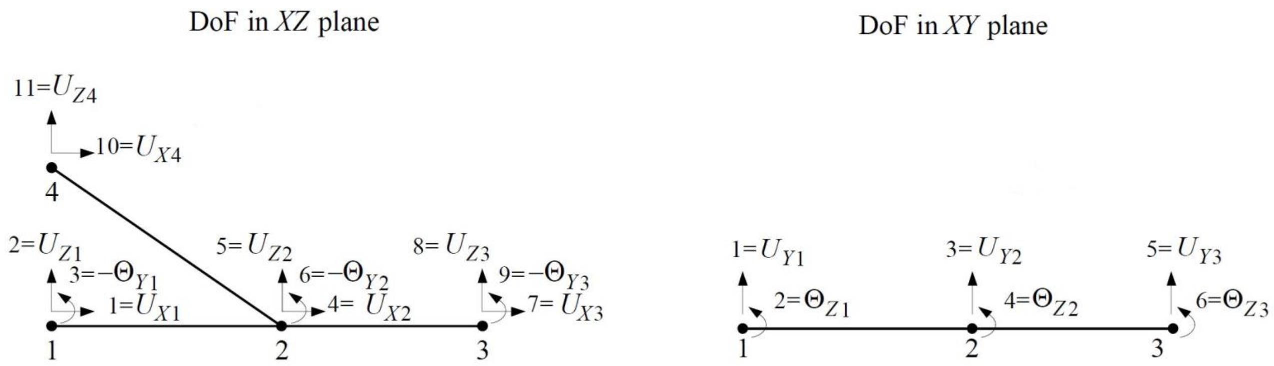

Figura 3: DOFs of the structure

DOFs in GLOBAL Reference Frame (X-Z plane)		u_{x1} 1	u_{z1} 2	$-\vartheta_{y1}$ 3	u_{x2} 4	u_{z2} 5	$-\vartheta_{y2}$ 6	u_{x3} 7	u_{z3} 8	$-\vartheta_{y3}$ 9	u_{x4} 10	u_{z4} 11
DOFs in LOCAL Reference Frame (X-Z plane)	EL I	u_{x1} 1	u_{z1} 2	$-\vartheta_{y1}$ 3	u_{x2} 4	u_{z2} 5	$-\vartheta_{y2}$ 6					
	EL II				u_{x1} 1	u_{z1} 2	$-\vartheta_{y1}$ 3	u_{x2} 4	u_{z2} 5	$-\vartheta_{y2}$ 6		
	EL III				u_{x1} 1	u_{z1} 2					u_{x2} 3	u_{z2} 4

DOFs in GLOBAL Reference Frame (X-Y plane)		u_{y1} 1	ϑ_{z1} 2	u_{y2} 3	ϑ_{z2} 4	u_{y3} 5	ϑ_{z3} 6
DOFs in LOCAL REference FRame (X-Y plane)	el.I	u_{y1} 1	ϑ_{z1} 2	u_{y2} 3	ϑ_{z2} 4		
	el.II			u_{y1} 1	ϑ_{z1} 2	u_{y2} 3	ϑ_{z2} 4

X-Z plane

The global stiffness matrix is 11-by-11 matrix, then the constraints are applied: the first three DOFs are not considered because of the clamp at the node 1 and the last two because of the hinge at node 4, so the active DOFs remain 6.

$$\begin{bmatrix} \\ & K & \\ & MATRIX & \\ \\ \end{bmatrix}_I \begin{Bmatrix} u_{x2} \\ u_{z2} \\ -\vartheta_{y2} \\ u_{x3} \\ u_{z3} \\ -\vartheta_{y3} \end{Bmatrix}$$

The only contribution to the mass matrix is given by the mass at node 3:

$$M_{XZ} \rightarrow \begin{bmatrix} & & & & & \\ & & & & & \\ & & & & & \\ & & & m & & \\ & & & & m & \\ & & & & & \end{bmatrix} \begin{Bmatrix} \ddot{u}_{x2} \\ \ddot{u}_{z2} \\ -\ddot{\vartheta}_{y2} \\ \ddot{u}_{x3} \\ \ddot{u}_{z3} \\ -\ddot{\vartheta}_{y3} \end{Bmatrix}$$

The global stiffness matrix is not band diagonal matrix because of the presence of the bar which is attached in the intermediate node 2.

X-Y plane

The global stiffness matrix is 6-by-6 matrix and then the first two DOFs are not considered because of the clamp at the node 1.

$$\begin{bmatrix} K \\ MATRIX \end{bmatrix}_{II} \begin{Bmatrix} u_{y2} \\ \vartheta_{z2} \\ u_{y3} \\ \vartheta_{z3} \end{Bmatrix}$$

The only contribute to the mass matrix is given by the mass at node 3:

$$M_{XY} \rightarrow \begin{bmatrix} & & & \\ & & & \\ & & m & \\ & & & \end{bmatrix} \begin{Bmatrix} \ddot{u}_{y2} \\ \ddot{\vartheta}_{z2} \\ \ddot{u}_{y3} \\ \ddot{\vartheta}_{z3} \end{Bmatrix}$$

Dynamic analysis of free response

X-Z plane

The computation of the natural frequencies is performed by the static reduction method. The two translational DOFs affected by the mass are the master DOFs: $\{q\}_m = \{u_{x3}, u_{z3}\}^T$, and the others are slave DOFs: $\{q\}_s = \{u_{x2}, u_{z2}, -\vartheta_{y2}, -\vartheta_{y3}\}^T$. It has to be noticed that the static reduction does not introduce any approximation because of the lumped inertial properties of the model. So, the method simply rearranges matrices and vectors putting master DOFs at the top: it possible to identify submatrices:

$$\begin{bmatrix} M_{11} & M_{12} \\ M_{21} & M_{22} \end{bmatrix} \begin{Bmatrix} \ddot{q}_m \\ \ddot{q}_s \end{Bmatrix} + \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{Bmatrix} q_m \\ q_s \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

Two matrix equations are obtained by developing the product:

$$[M_{11}]\{\ddot{q}_m\} + [K_{11}]\{q_m\} + [K_{12}]\{q_s\} = \{0\}$$

$$[K_{12}]\{q_m\} + [K_{22}]\{q_s\} = \{0\}$$

Computing $\{q_s\}$ by one equation and substituting it in the other one a 2-by-2 eigenvalue problem is given:

$$\{-\lambda_n^2 [M]_{cond} + [K]_{cond}\}\begin{Bmatrix} u_{x30} \\ u_{z30} \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}$$

where $[M]_{cond} = [M_{11}]_{2x2}$ and $[K]_{cond} = [K_{11}]_{2x2} - [K_{12}]_{2x4}[K_{22}]_{2x4}^{-1}[K_{21}]_{4x2}$

Two eigenvectors and two natural frequencies can be computed.

λ_{n2} **=673.10 rad/s**

λ_{n3}**=3814.60 rad/s**

The eigenvectors only contain the modal displacements of the master nodes. It is necessary to compute the modal displacements of the slaves:

$$\{q_s\} = -([K_{22}]^{-1}[K_{21}])\{q_m\}$$

$\{q_m\}$ and $\{q_s\}$are collected in the modal displacement vectors and its elements are sorted coherently with the vector of generalized coordinates.

The mode-shapes are listed below:

$$\begin{Bmatrix} u_{x2} \\ u_{z2} \\ -\vartheta_{y2} \\ u_{x3} \\ u_{z3} \\ -\vartheta_{y3} \end{Bmatrix}_2 = \begin{Bmatrix} 0.1754 \\ 0.0085 \\ -0.0677 \\ 0.2581 \\ -0.0072 \\ -0.1583 \end{Bmatrix} \qquad \begin{Bmatrix} u_{x2} \\ u_{z2} \\ -\vartheta_{y2} \\ u_{x3} \\ u_{z3} \\ -\vartheta_{y3} \end{Bmatrix}_3 = \begin{Bmatrix} 0.0071 \\ 0.1453 \\ 0.8496 \\ 0.0072 \\ 0.2581 \\ 0.9505 \end{Bmatrix}$$

X-Y plane

In X-Y plane only one master DOF is present

$$\{q\}_m = \{u_{y3}\}^T, \{q\}_s = \{\vartheta_{z2}, u_{y2}, \vartheta_{z3}\}^T$$

Following the same procedure explained in previous point it is possible to obtain single DOF eigenproblem

$$(-\lambda_n^2 m_{cond} + k_{cond})\{u_{y3}\} = \{0\}$$

where m_{cond} and k_{cond} are scalar.

The eigenvector and natural frequency are computed.

λ_{n1} **=589.59 rad/s**

The modal displacement vector is computed as in X-Y plane:

$$\begin{Bmatrix} u_{y2} \\ \vartheta_{z2} \\ u_{y3} \\ \vartheta_{z3} \end{Bmatrix}_1 = \begin{Bmatrix} 0.1488 \\ 0.8382 \\ 0.2582 \\ 0.9156 \end{Bmatrix}$$

Dynamic response to the excitation

The system is excited in two different ways: motion of the constraints with harmonic oscillation and shock excitation. The aim is to perform a strength analysis of the structure.

Motion of the constraints with harmonic oscillation

In the harmonic case, different inputs are considered depending on the forcing frequency:

Harmonic oscillation simultaneous along X,Y and Z	
Frequency λ_e (Hz)	Intensity
5÷8.5	ampl. 10 mm
8.5÷35	acc. 3g
35÷50	acc. 1g

The harmonic oscillations are applied at nodes 1 and 4 along X, Y and Z axes. The dynamic analysis is performed separately in X-Z and X-Y plane and leads to the nodal displacements.

X-Z plane

The DOFs vector is $\{q\}$ so the system is described by the following equation of motion:

$$[M]_{xz}\{\ddot{q}\} + [K]_{xz}\{q\} = -[M]_{xz}\{\delta_x\}\ddot{x}_A - [M]_{xz}\{\delta_z\}\ddot{z}_A$$

where $\{\delta_x\}$ and $\{\delta_z\}$are $\{\delta_x\} = \{1 \quad 0 \quad 0 \quad 1 \quad 0 \quad 0\}^T$, $\{\delta_z\} = \{0 \quad 1 \quad 0 \quad 0 \quad 1 \quad 0\}^T$ which allow to decompose the contribution of the forcing input in X and Z direction. In order to perform modal analysis, modal coordinates are introduced and the eigenvectors are normalized with respect to the modal mass component $\sqrt{m}$. It is possible to express equations of motion as shown below:

$$[\bar{\phi}]^T [M]_{xz} [\bar{\phi}] \begin{Bmatrix} \ddot{\eta}_2 \\ \ddot{\eta}_3 \end{Bmatrix} + [\bar{\phi}]^T [K]_{xz} [\bar{\phi}] \begin{Bmatrix} \eta_2 \\ \eta_3 \end{Bmatrix} = -[\bar{\phi}]^T [M]_{xz} \{\delta_x\}\ddot{x}_A - [\bar{\phi}]^T [M]_{xz} \{\delta_z\}\ddot{z}_A$$

The response is of the same type of the forcing function:

$$\begin{Bmatrix} x_A \\ z_A \end{Bmatrix} = \begin{Bmatrix} x_{A0} \\ z_{A0} \end{Bmatrix} e^{i\lambda_e t}$$

in which λ_e is the frequency of the forcing function. By computing the derivatives with the respect to time and substituting in the equation of motion in modal coordinates, modal displacements can be obtained:

$$\eta_{02} = \frac{\lambda_e^2 \left\{ \left[\bar{\phi}_2\right]^T \left[M\right]_{xz} \left\{\delta_x\right\} x_{A0} - \left[\bar{\phi}_2\right]^T \left[M\right]_{xz} \left\{\delta_z\right\} z_{A0} \right\}}{\lambda_{n2}^2 - \lambda_e^2}$$

$$\eta_{03} = \frac{\lambda_e^2 \left\{ \left[\bar{\phi}_3\right]^T \left[M\right]_{xz} \left\{\delta_x\right\} x_{A0} - \left[\bar{\phi}_3\right]^T \left[M\right]_{xz} \left\{\delta_z\right\} z_{A0} \right\}}{\lambda_{n3}^2 - \lambda_e^2}$$

In the last two ranges of frequencies the intensity of the harmonic is given in term of acceleration and so x_{A0} and z_{A0} are computed:

$$x_{A0} = z_{A0} = \frac{3g}{\lambda_e^2}, \; x_{A0} = z_{A0} = \frac{g}{\lambda_e^2}$$

It is possible to calculate physical displacement:

$$\{q\} = \begin{Bmatrix} u_{x2} \\ u_{z2} \\ -\vartheta_{y2} \\ u_{x3} \\ u_{z3} \\ -\vartheta_{y3} \end{Bmatrix} = \left[\bar{\phi}\right] \begin{Bmatrix} \eta_{02} \\ \eta_{03} \end{Bmatrix}$$

X-Y plane

Following the same procedure previously explained it is possible to obtain modal displacement:

$$\eta_{01} = \frac{\lambda_e^2 \left[\bar{\phi}_1\right]^T \left[M\right]_{xy} \left\{\delta_y\right\} x_{y0}}{\lambda_{n1}^2 - \lambda_e^2}$$

and the vector of physical displacement is:

$$\{q\} = \begin{Bmatrix} u_{y2} \\ \vartheta_{z2} \\ u_{y3} \\ \vartheta_{z3} \end{Bmatrix} = \left\{\bar{\phi}\right\}_{4x1} \eta_{01}$$

The computation has been performed for 5-80Hz frequencies range, with 0.1Hz pace.

Response to shock excitation

The response of the system to the shock forcing input is investigated in perpendicular direction to X-Z plane because the system in this direction has not the stiffening contribution of the bar. Because the shock is due to acceleration of the constraints, the equation of motion is:

$$\left[\bar{\phi}_1\right]^T \left[M\right]_{xy} \left[\bar{\phi}_1\right] \left\{\ddot{\eta}_1\right\} + \left[\bar{\phi}_1\right]^T \left[K\right]_{xy} \left[\bar{\phi}_1\right] \left\{\eta_1\right\} = -\left[\bar{\phi}_1\right]^T \left[M\right] \left\{\delta_y\right\} \ddot{y}_A$$

where $\{\delta_y\}$ as in previous case allows to identify the contribution of the forcing term at proper direction Y.

It Is possible to rearrange previous equation:

$$\ddot{\eta}_1 + \lambda_{n1}^2 \eta_1 = -[\bar{\phi}_1]^T [M]_{xy} \{\delta_y\} \ddot{y}_A$$

In which $\ddot{y}_A$ is the acceleration of the constraints modelled with ramp input signal with slope of 20g/0.011s, and integrate it numerically through the following Simulink model:

The output of the model is the value of η_1 at each integration time step and by the eq. $\{q\} = [\bar{\phi}]\{\eta\}$ the

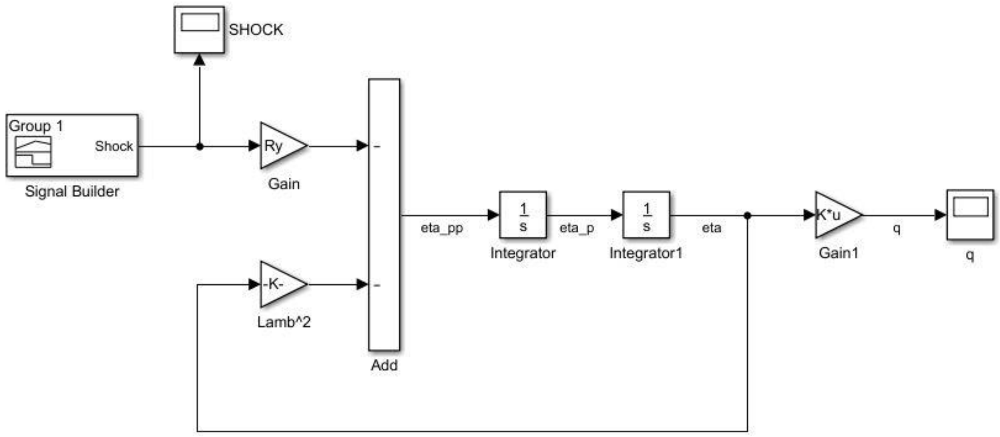

Figure 4: Simulink model

physical displacements in time of the node 2 and 3 are obtained. In order to have a more accurate result a fixed-step integrator is chosen and, considering that the impulse is applied in 11ms, the time step is fixed 0,01ms.

Strength analysis

The verification is made for both harmonic and shock excitation as described below.

Harmonic oscillation simultaneous along X,Y and Z		Shock along Y
Frequency λ_e (Hz)	Intensity	Intensity
5÷8.5 8.5÷35 35÷50	ampl. 10 mm acc. 3g acc. 1g	acc. from 0 to 20g in 11 ms with linear increment

The stress analysis is made only on the element I, because it has the contributions of the bending and axial stresses, at node 1 and 2.

Harmonic oscillation

It is requested to make resistance verification of the structure with respect to the MIL-STD 810C as summarized in the following table:

Forced response	
Harmonic oscillation simultaneous along X,Y and Z	Shock along Y
$\max(U_{X3}, U_{Y3}, U_{Z3}) \leq 3$ mm $\sigma \leq R_m/1.575$ $\sigma \leq R_{p0.2}/1.155$ $\sigma \leq \sigma_{D-1}$	$U_{Y3} \leq 3$ mm $\sigma \leq R_m$

In both X-Z plane and X-Y plane the displacements at the node 3 computed in the previous section satisfy the verification requested in the table.

	Max Displacement [m]
u_x	$4.06 \cdot 10^{-6}$
u_y	$9,82 \cdot 10^{-5}$
u_z	$8,46 \cdot 10^{-5}$

In both the X-Z and X-Y planes the vectors of the reaction forces for each element I has been calculated by:

$$\{F\} = [K]\{q\}$$

The maximum stress in any section of the beam has three harmonic contributions: bending due to My and Mz and normal stresses.

$$\sigma_{\max} = \sigma_{bendXZ} + \sigma_{bendXY} + \sigma_N$$

This calculation is made for each frequency and the maximum value is extracted.

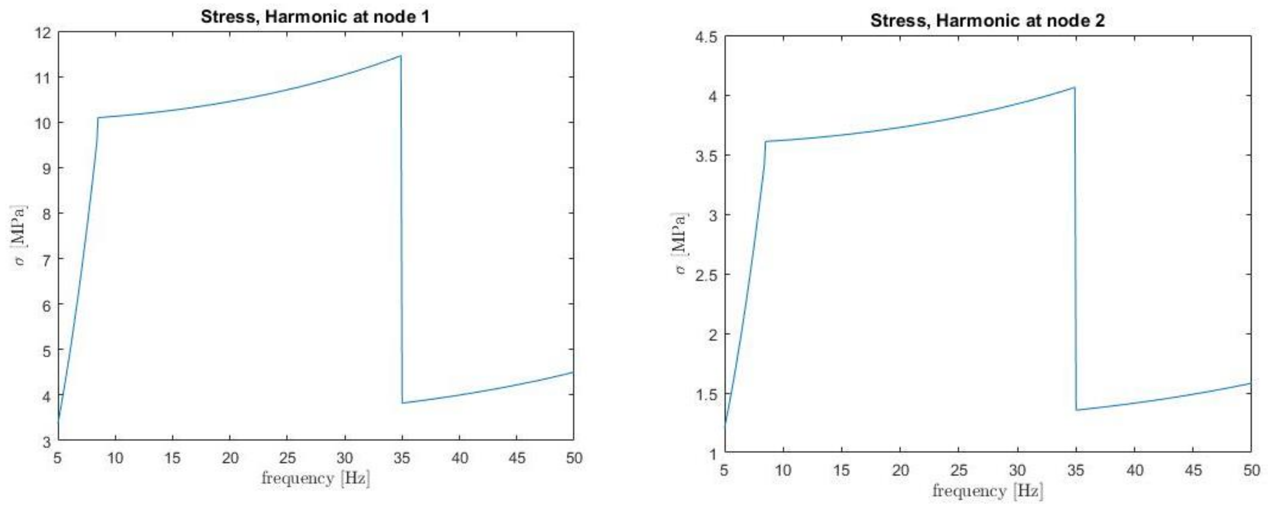

Figure 5: Stresses due to the harmonic excitation

The more critical condition occurs at node 1, due to the bending behavior in x-y plane, in which the maximum stress is 10.97 MPa. At this node, the comparison between the maximum value and the $R_{p0,2}$ and R_m satisfies the requests. The same thing occurs for fatigue verification.

Shock along Y

The maximum displacement along the y axis is 0.55mm and so the request is satisfied. The stress in any section of the beam I is the one due to the M_z and it is computed for each time step of the integration.

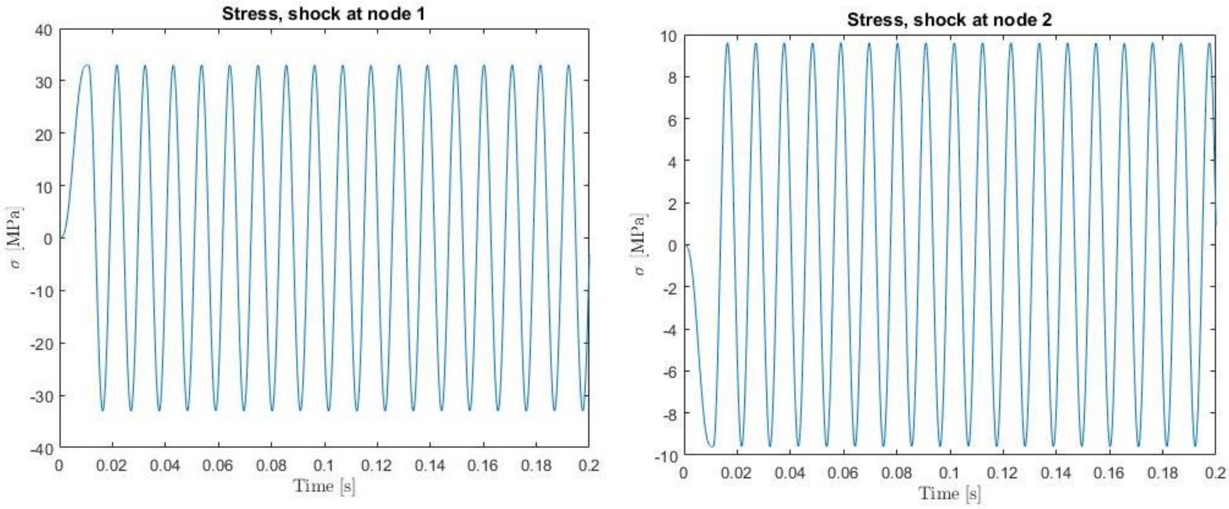

Figure 6:Stresses due to the shock excitation

Then the maximum value is extracted and the comparison of the resulting stress is made only with respect to R_m. In fact, the shock models a rare event, so the comparison with $R_{p0,2}$ is not take into account because in this case small yield is acceptable. The maximum stress is located, also in this case, at node 1 and it is equal to 33,01 MPa, so the request is verified.