



FINITE ELEMENT METHOD

AEF-A.2.13

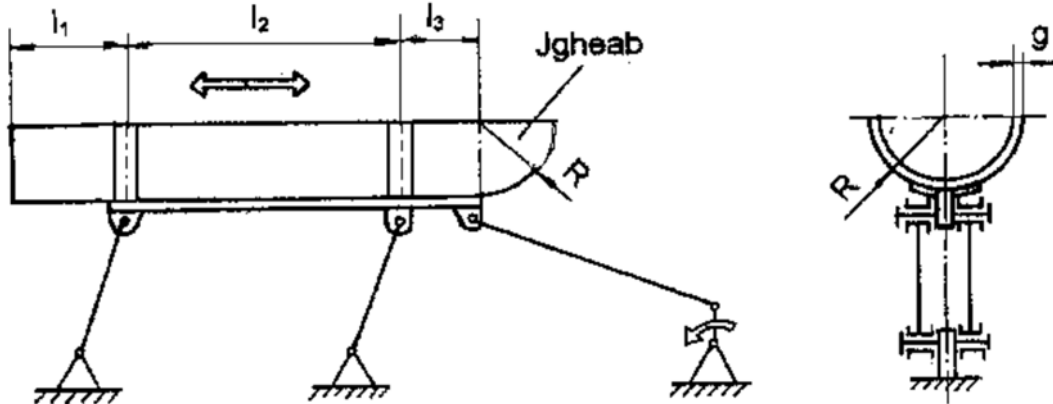
NATURAL FREQUENCIES AND VIBRATION ANALYSIS OF MEMBRANES

INTRODUCTION

In many practical situations for the design of complex mechanical systems it is necessary to know the natural frequencies and the vibration modes of some components or even the assembly. These parameters, invariant with time, determined under the conditions of respecting the equilibrium configuration, are intimate characteristics of the structure dependent on shape, size and material.

DESCRIPTION OF THE APPLICATION

The design of the vibrating feed machine in the adjacent figure involves studying the natural frequencies and its own oscillation shapes of the semicircular trough made of composite material, Kevlar 49 Epoxy. The structure of the trough is stiffened in the area of the support and drive levers by including suitable metal parts. The natural frequencies obtained by analyzing and solving the model with finite elements are useful in design for the study of resonance phenomena and dynamic (modal) analysis of the structure. Because with FEM, continuous structures are approximated by a finite element system, and their own value analyzes lead to approximate solutions that can be minimized by optimal meshing.



PURPOSE OF THE APPLICATION

In this application we present the study of the natural frequencies and vibration modes for the vibrator gutter subassembly above. The dimensions of the trough in the middle layer of the layered membrane are: $l_1 = 450$ mm, $l_2 = 890$ mm, $l_3 = 300$ mm and $R = 210$ mm. The reinforcing reinforcement made of OL37 steel has the dimensions $R_1 = 210$ mm, $b = 100$ mm and thickness $g_1 = 4$ mm.

The indices of the characteristic parameters of the material are the same as the notations of the coordinate system directions notations associated with the material (1 and 2 - directions in the orthotropic plane and normal direction of this plane). The structure of the gutter is composed of four symmetrical structures, each 0.5 mm thick and with the main directions $90 / -90 / 45 / -45$.

Application AEF-A.2.13

ANALYSIS OF NATURAL FREQUENCIES AND VIBRATION MODES OF MEMBRANE STRUCTURES

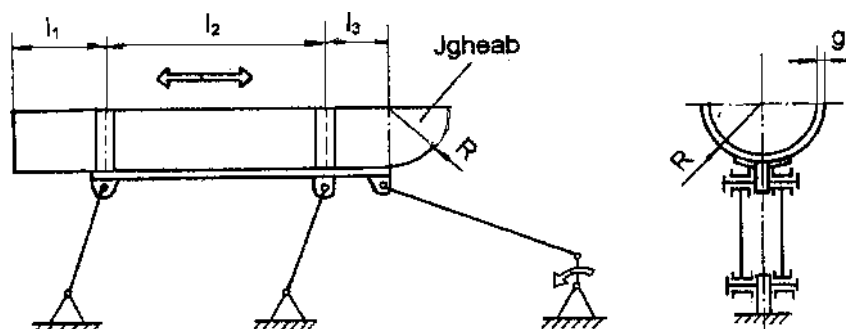
A. DEFINITION OF THE APPLICATION

A.1. Introduction

In many practical situations for the design of complex mechanical systems, it is necessary to know the natural frequencies and vibration modes of some components or even the entire assembly. These parameters, invariant with time and determined under conditions respecting the equilibrium configuration, are intimate characteristics of the analyzed structure dependent on shape, dimensions, and material.

A.2. Description of the application

The design of the vibrating feed machine in the adjacent figure involves studying the natural frequencies and the eigenmodes of oscillation of the semicircular trough (gutter) made of composite material, Kevlar 49 Epoxy. The structure of the trough is stiffened in the area of the support and drive levers by including metal parts with appropriate shapes. The natural frequencies obtained by analyzing and solving the finite element model are useful in design for the study of resonance phenomena and for the dynamic (modal) analysis of the structure. Because FEM approximates continuous structures with a finite element system, eigenvalue analyses lead to approximate solutions that can be minimized through optimal meshing.



A.3. Purpose of the application

This application presents the study of natural frequencies and vibration modes for the vibrator trough subassembly shown above. The dimensions of the trough at the level of the middle fibers of the layered membrane are: $l_1 = 450$ mm, $l_2 = 890$ mm, $l_3 = 300$ mm and $R = 210$ mm. The stiffening reinforcement made of OL37 steel has the dimensions $R_1 = 210$ mm, $b = 100$ mm and thickness $g_1 = 4$ mm.

The indices of the characteristic material parameters are the same as the notations of the coordinate system directions associated with the material (1 and 2 are directions in the orthotropy plane and z is the direction normal to this plane). The trough structure is composed of four symmetrical structures, each 0.5 mm thick and with main directions 90/-90/45/-45.

B. PREPARATION OF THE ANALYSIS MODEL

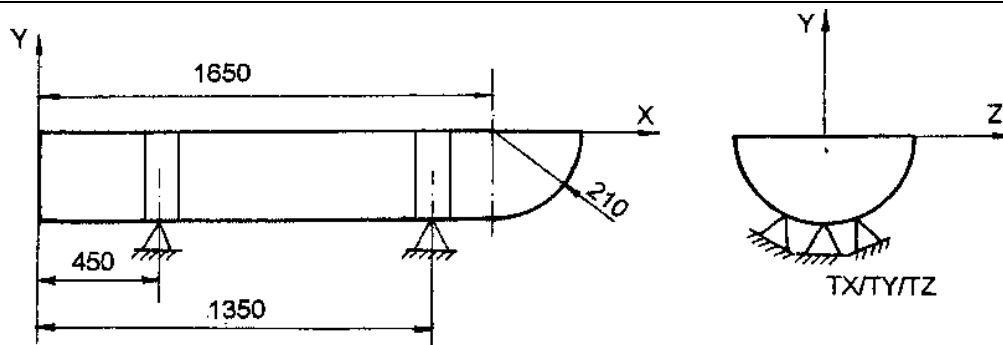
B.1. Defining the analysis model

To prepare the finite element analysis model associated with the application above, the following must be identified:

- Geometric shape and dimensions.
- Restrictions induced by connections with adjacent elements.
- Material characteristics.

B.2. Description of the analysis model

The geometric shape and dimensions of the analysis model for the trough and stiffening reinforcements are presented in the figure below. Since the trough supports have high stiffness, the analysis model structure is considered supported in the areas where it is fixed to them, by canceling translation displacements relative to the global XYZ system. Finite element analysis of natural frequencies and eigenmodes of an elastic structure, being an eigenvalue problem, does not imply loading it.



B.3. Establishing material characteristics

For the finite element analysis, the material characteristics are:

- For OL 37 steel:
 - Longitudinal modulus of elasticity (Young's Modulus) $E = 204000 \text{ N/mm}^2$
 - Transverse contraction coefficient (Poisson's ratio): $\nu = 0,3$
- For Kevlar 49 Epoxy composite material:
 - Longitudinal moduli of elasticity: $E1 = 1,0701 \times 10^7 \text{ MPa}$, $E2 = 5,4375 \times 10^5 \text{ MPa}$.
 - Transverse moduli of elasticity: $G12 = 2,523 \times 10^5 \text{ MPa}$, $G1z = 2,523 \times 10^5 \text{ MPa}$, $G2z = 2,523 \times 10^5 \text{ MPa}$
 - Transverse contraction coefficient: $\nu = 0,4$
 - Density: $\rho = 2,6 \times 10^{-6} \text{ kg/mm}^3$

C. PRE-PROCESSING OF THE ANALYSIS MODEL

C.1. Geometric modeling

C.1.1. Geometric modeling of the trough

Activating the solid generation module and setting units

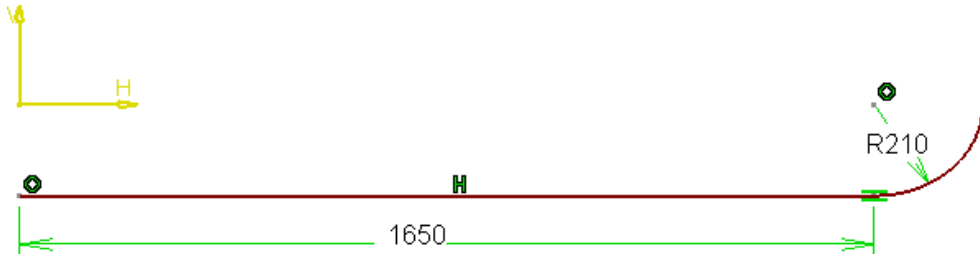
Start → Mechanical Design → Part Design → New part: New part name: Jgheab.

Tools → **Options...** → **Options**: Parameters and Measure; Units; Length, Milimeter (mm); **OK**.

Activating the shape generation module and generating the reference sketch

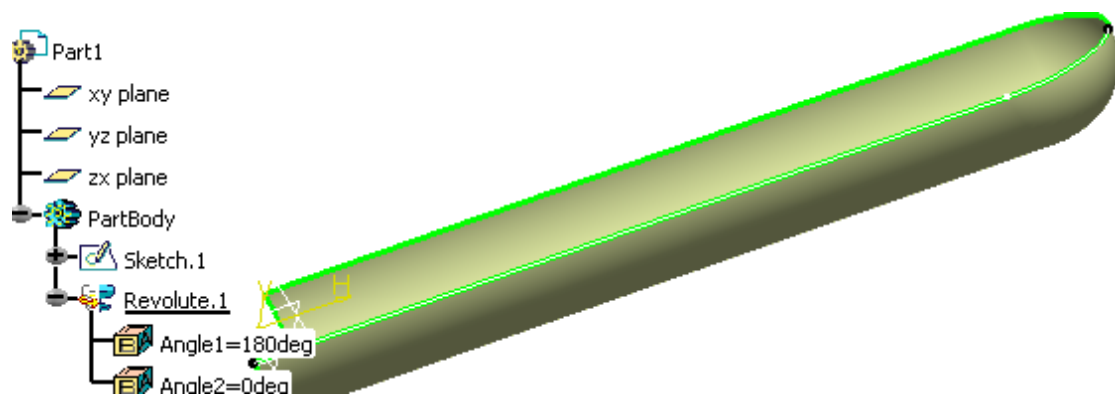
Start → **Shape** → **Generative Shape Design**.

(Sketcher) → xy plane → (Profile) → [draw the profile from the sketch below with the mouse] → (Constraint) [enter dimensions successively by selecting entities] → (Exit Workbench).



Generating the trough body

(Revolve) → **Revolution Surface Definition**: Angle 1: 180deg, Profile Selection [select the reference sketch, **Sketch.1** with the mouse], **OK**.



Introducing material characteristics for the trough

[Select **PartBody** Jgheab] → (Apply Material) → **Library (ReadOnly)** Metal, Steel, Apply Material; **OK**.

[Select **Steel** specification in the tree] → **Properties**: Feature Properties, Feature Name: Kevlar 49 Epoxy; Analysis, Material: Fiber Material; Longitudinal Young Modulus $1,0701 \times 10^7$ MPa, Transverse Young Modulus $5,4375 \times 10^5$ MPa, Poisson Ratio 0.4, Shear Modulus in XY Plane: $= 2,523 \times 10^5$ MPa, Shear Modulus in YZ Plane: $= 2,523 \times 10^5$ MPa, Density: 2600 kg/m^3 ; Apply; **OK**.

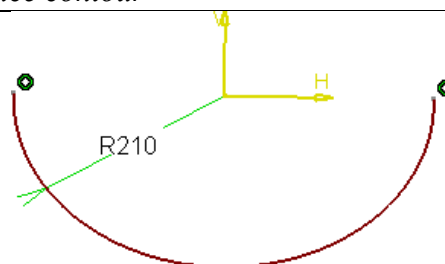


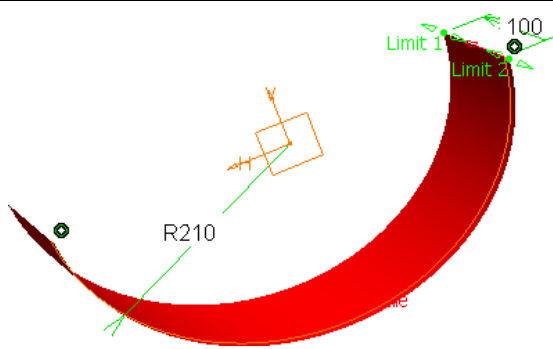
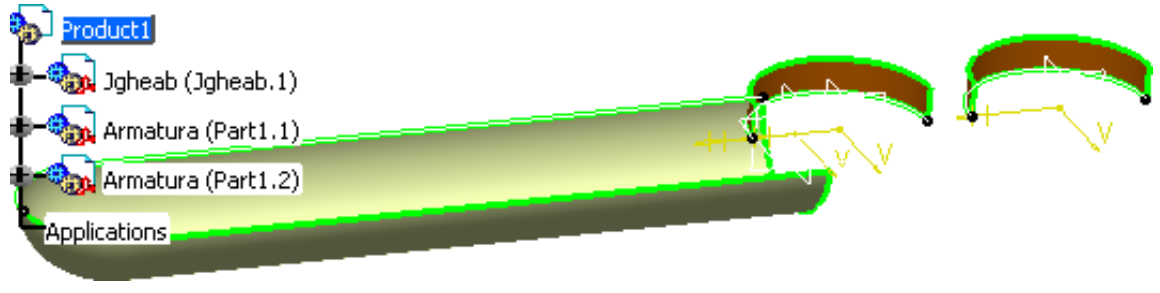
C.1.2 Geometric modeling of the reinforcement

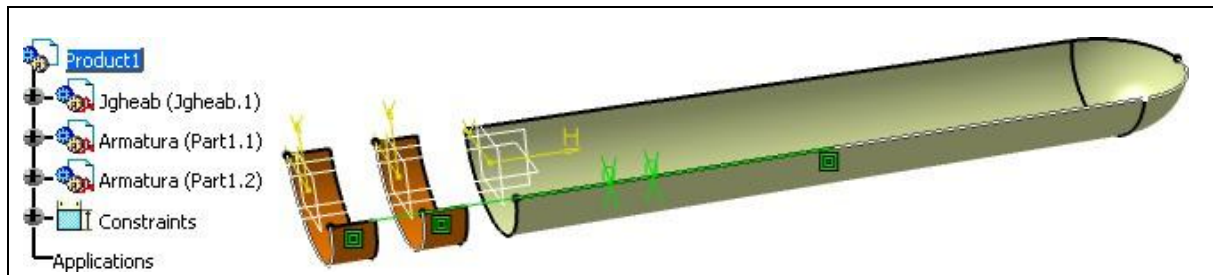
Generating the reference contour

Start → **Shape** → **Generative Shape Design**.

(Sketcher) → xy plane → (Arc) → [draw a semicircle] → (Constraint) [select the semicircle and enter the radius] → (Exit Workbench).

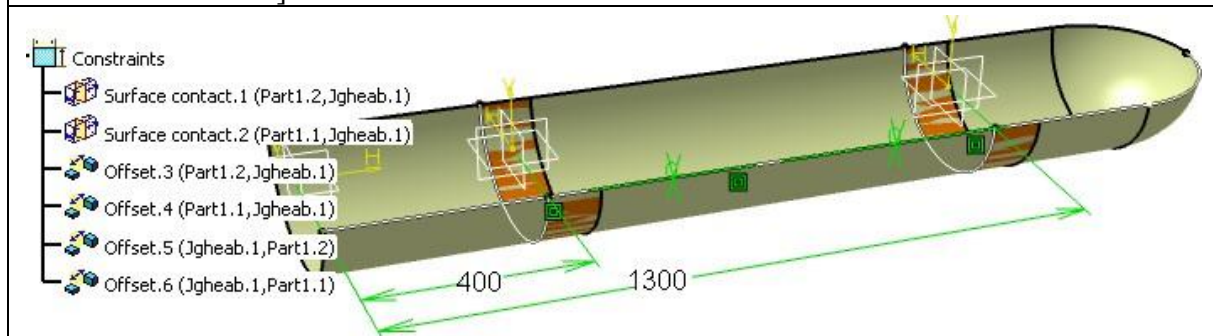


Generating the reinforcement	
 (Extrude) → Extrude Surface Definition: Profile: [select Sketch.1], Direction: XY Plane, Extrusion Limits, Limits.1, Type: Dimension, Dimension: 100 mm, ↵OK.	
Introducing material characteristics for the reinforcement	
[Select PartBody specification] →  (Apply Material) → Library (ReadOnly): Metal, Steel, Apply Material. ↵OK. [Select Steel specification in the tree] → Properties: Feature Properties, Feature Name: OL37; Analysis, Young Modulus 204000 N/mm ² , Poisson Ratio 0,3; Apply; ↵OK.	
C.2. Geometric modeling of the assembly	
Activating the assembly module and loading components	
Start → Mechanical Design → Assembly Design. Insert → Existing Component → [activate Product1] → [select path] → Jgheab.CATPart → Open. [Repeat to load the file Armatura.CATPart twice].  (Manipulation) → Manipulation Parameters: [select direction, move body to avoid overlap], OK.	
	
Introducing assembly geometric constraints	
[Select with the mouse the coincident cylindrical surfaces on the Gutter and on the Reinforcement, using the Ctrl key] →  (<i>Contact Constraint</i>) → Constraint Properties: Name: Surface contact.1,  Surface contact ↵OK. [Repeat this sequence for the other set of common cylindrical surfaces Gutter - Reinforcement] [Select with the mouse the edges corresponding to the OX axis on the Gutter and on the Reinforcement, using the Ctrl key] →  (<i>Offset Constraint</i>) → Constraint Properties: Name: Offset.3,  Parallel axis, Offset: 0 mm, ↵OK →  (Update All). [Repeat this sequence for the positioning of the other reinforcement].	



[Select with the mouse the XOY plane corresponding to the reference system of a reinforcement, respectively the YOZ plane on the Gutter, using the Ctrl key] → (Offset Constraint)

→ **Constraint Properties:** Name: Offset.5, Orientation: Same, Offset: 1300 mm, ↵OK → (Update All). [This sequence is repeated for positioning the other reinforcement, at a distance of 400 mm].



C.3. Finite element modeling

Activating the advanced meshing module

Start → Analysis & Simulation → Advanced Meshing Tools → **New Analysis Case:** Static Analysis, ↵OK.

Introducing element type and meshing characteristics



(Surface Mesher) → [select surface **Jgheab.1**] → **Global Parameters:** Shape, Type: Linear, Mesh size: 50 mm, ↵OK → (Exit Workbench).

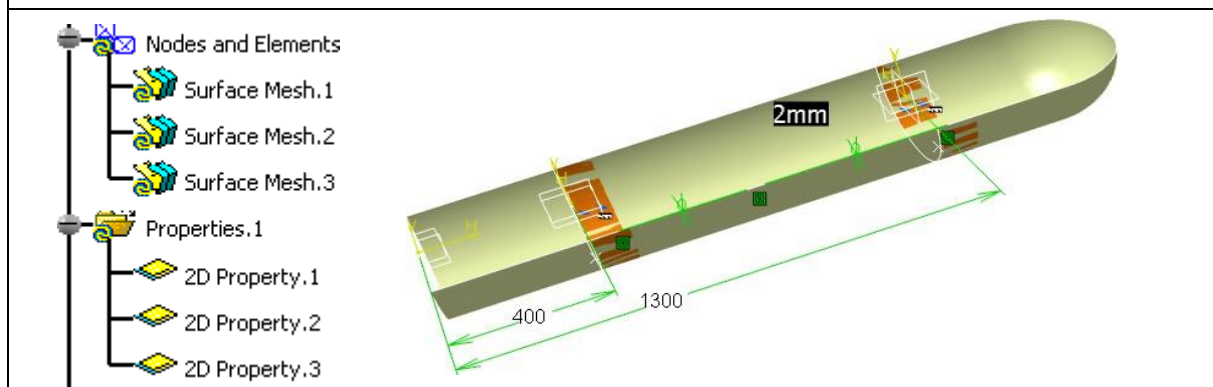
[This sequence is repeated for the two Reinforcement pieces, choosing the finite element size (Mesh size) of 25 mm.].

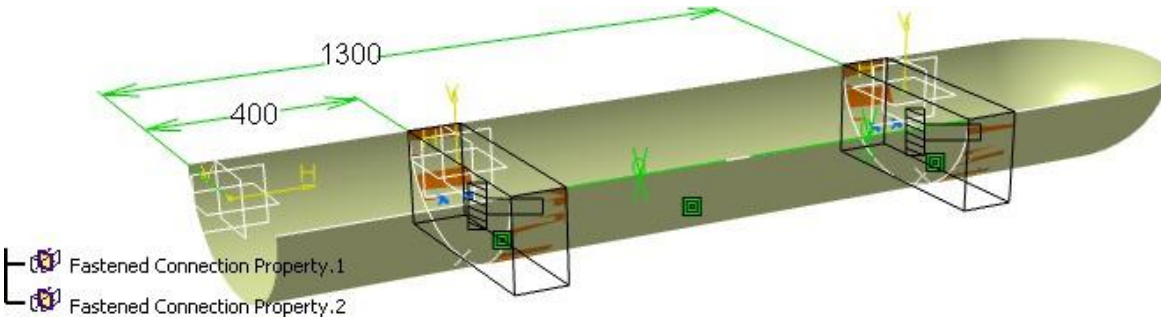
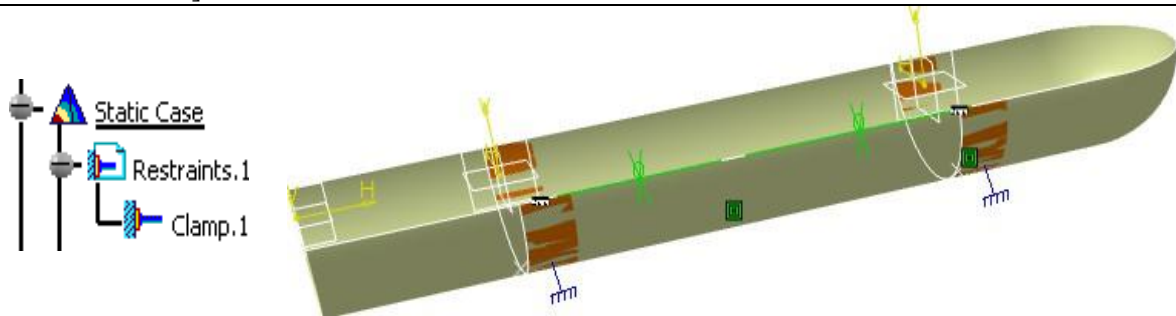
Activating the analysis module and properties

Start → Analysis & Simulation → Generative Structural Analysis.



(2D Property) → [select surface **Jgheab.1**] → **2D Property:** Thickness: 2 mm, ↵OK. [Repeat this sequence for the two Reinforcement pieces, entering a Thickness of 4 mm.].



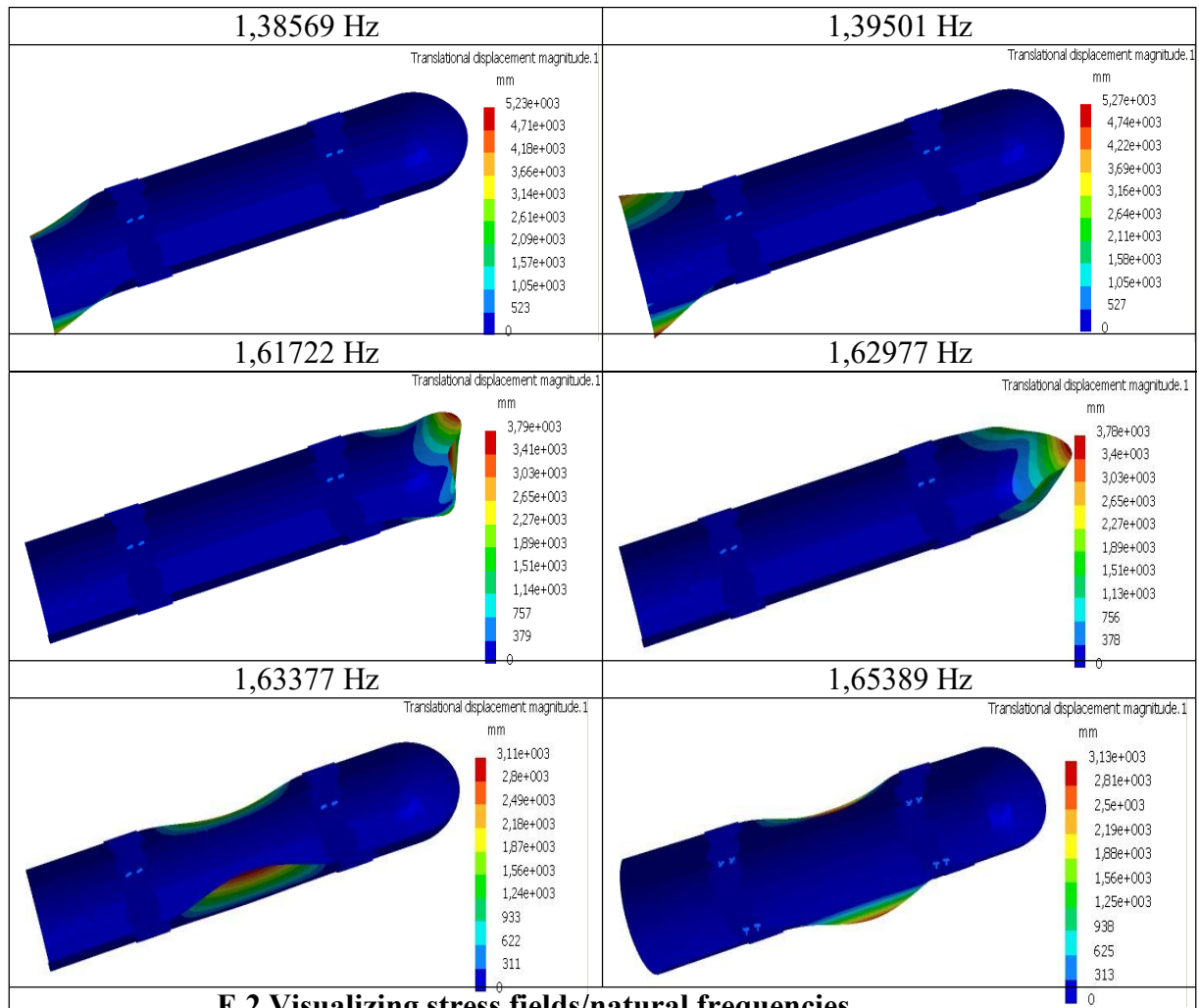
C.4. Modeling constraints	
<i>Modeling surface-to-surface connections</i>	
(Fastened Connection) → Fastened Connection : Supports: [select geometric restriction Surface Contact.1(Part1.2, Jgheab.1)], ↵OK. [Repeat for Surface Contact.2].	
	
<i>Introducing displacement restrictions</i>	
(Clamp) → Clamp Name Clamp.1, Supports [select outer surfaces of the reinforcements], ↵OK.	
	

D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the calculation module	
(Compute) → Compute : ↵ All, ↵OK → Computation Resources Estimation , ↵Yes → Computation Status .	
D.2. Activating the natural frequency analysis module	
Insert → F requency Case → F requency Case: ☒ Restraint, ☐ Masses, ☐ Static Case Solution, ☒ Hide existing analysis case, ↵OK.	
D.3. Re-launching the calculation module	
(Compute) → Compute : ↵ All, ↵OK → Computation Resources Estimation , ↵Yes → Computation Status .	

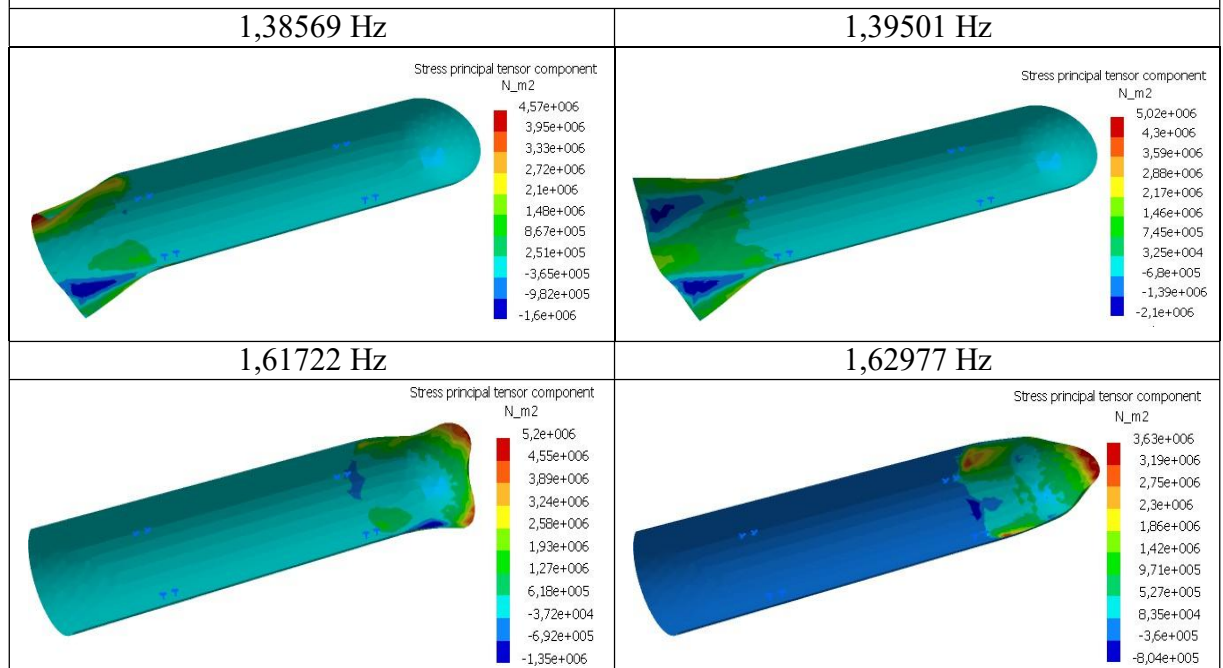
E. POST-PROCESSING OF RESULTS

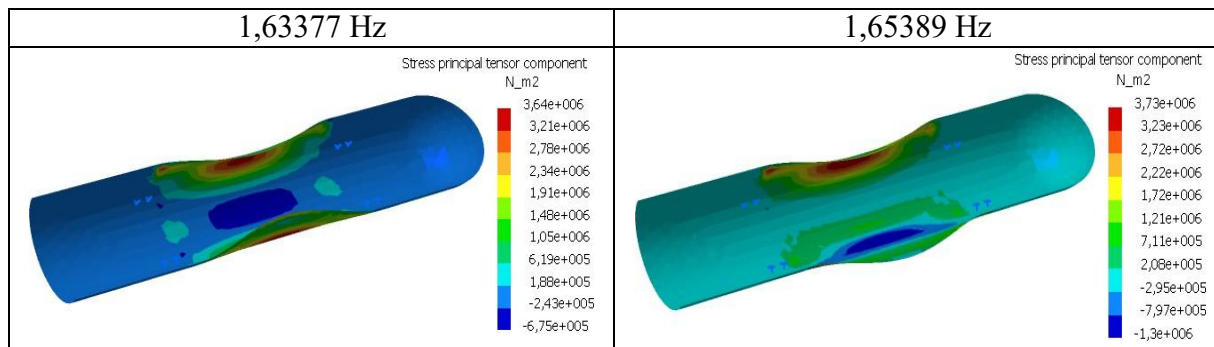
E.1. Visualizing displacement fields/natural frequencies	
(Displacement) → [mouse selection, right button, in the image area with the displacements represented with vectors] → Translational displacement vector.1 object → Definition → I mage E dition: Visu: Average iso; Occurrence: Frequency (Hz) [select frequency value from list], ↵OK.	



E.2 Visualizing stress fields/natural frequencies

(Principal Stress) → [right-click on vector image] → Stress principal tensor symbol.1
object → Definition → **Image Edition** → Visu: Average iso; Occurrences: Frequency (Hz)
[select natural frequency value from the list of values],
↵OK.





F. CONCLUSIONS

Determining the natural frequencies and corresponding vibration modes of mechanical system elements using high-performance programs based on FEM is the only viable option. This possibility is necessitated by the complex geometric shapes and the diverse connections of these elements with other parts of the assembly.

The data obtained from the finite element analysis of vibration frequencies and modes (natural frequencies, vibration shapes, displacements, stresses, reactions, etc.) are useful in design activity to avoid system operation at resonance or can be used as data input for modal analyses, also using high-performance FEM programs, to study dynamic behavior. For the analyzed trough, **10 natural frequencies** were identified: **1.38569 Hz, 1.39501 Hz, 1.61722 Hz, 1.62977 Hz, 1.63377 Hz, 1.65389 Hz, 2.88587 Hz, 2.89281 Hz, 2.92392 Hz, 3.47534 Hz**. If more natural frequencies need to be identified, the desired number is set in the **Image Edition** window, **Occurrences** tab, accessible by right clicking the image in the workspace. The images above present the displacement and stress fields for the first 6 eigenmodes.

G. EXERCISE

Analyze, using finite elements, the natural frequencies and corresponding vibration modes of the **corrugated tube** component of the flexible coupling shown in the adjacent figure. This element is made of **stainless steel** and has dimensions: $r = 10$ mm, $p = 20$ mm, $D = 50$ mm, $g = 0,3$ mm.

