



FINITE ELEMENT METHOD

AEF-A.2.14

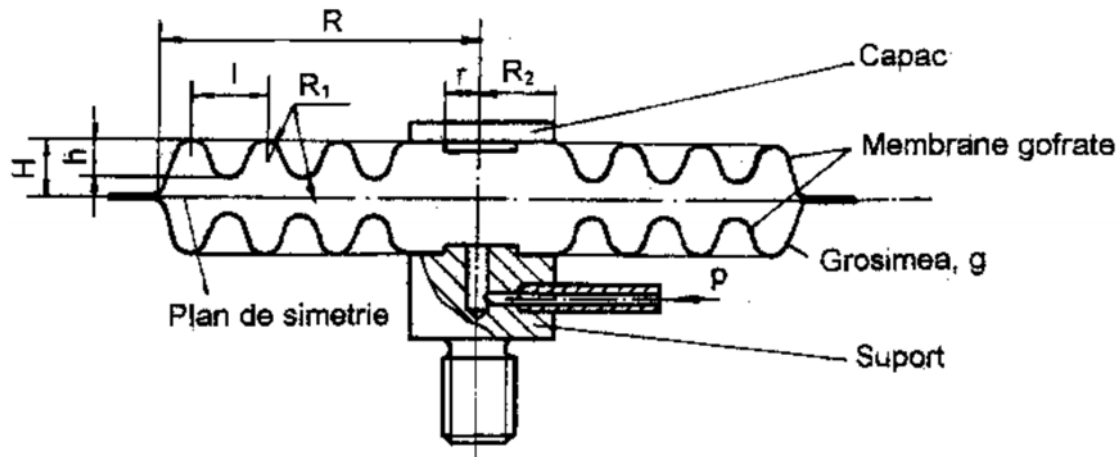
DYNAMIC ANALYSIS OF MEMBRANES

INTRODUCTION

Mechanical systems sometimes contain two-dimensional elements with a much reduced thickness, usually called membranes. Their structures during operation may be characterized by increased displacements with significant changes in load directions, usually statically applied (without considering time variation), or may be dynamically required when determining their response to certain loading regimes in time. The dynamic analysis problems of these MEF-based performance-based structures for resolving use appropriate modules.

DESCRIPTION OF THE APPLICATION

Elastic membranes, usually of circular shape, may be components of precision manometers (altimeters, speedometers, vacuums, etc.) with wide fluid pressure ranges. Also, the elastic membranes can be used as pressure transducers, force or displacements that usually engage with other transducers such as piezoelectric or magneto elastic. In systems where the supply fluid pressure is integrated, it has variable values. The dynamic response of the membrane structure to the action of the feed pressure has direct implications on the measurement accuracy of the output parameter.



PURPOSE OF THE APPLICATION

In this application, linear dynamic analysis is performed to determine the dynamic response of the embossed elastic membrane structure in the manometer caps of a dashboard shown in the figure above. The two corrugated membranes of the manometry capsule, made of stainless steel with the elastic modulus $E = 1,8 \cdot 10^5 \text{ N / mm}^2$ and the cross-shrinkage coefficient $\nu = 0,3$, have the dimensions $R = 48 \text{ mm}$, $r = 5 \text{ mm}$, $R_1 = 0.3 \text{ mm}$, $l = 10 \text{ mm}$, $R_2 = 10 \text{ mm}$, $h = 2 \text{ mm}$, $H = 2.5 \text{ mm}$.

Supply pressure is the variation shown in the figure below. In order to take into account the membrane driven subsystem, materialized in the above figure by means of a lid, a distributed mass on the flat surface is introduced, $m = 0.05 \text{ kg}$.

Application AEF-A.2.14

DYNAMIC ANALYSIS OF MEMBRANE-TYPE MECHANICAL STRUCTURES

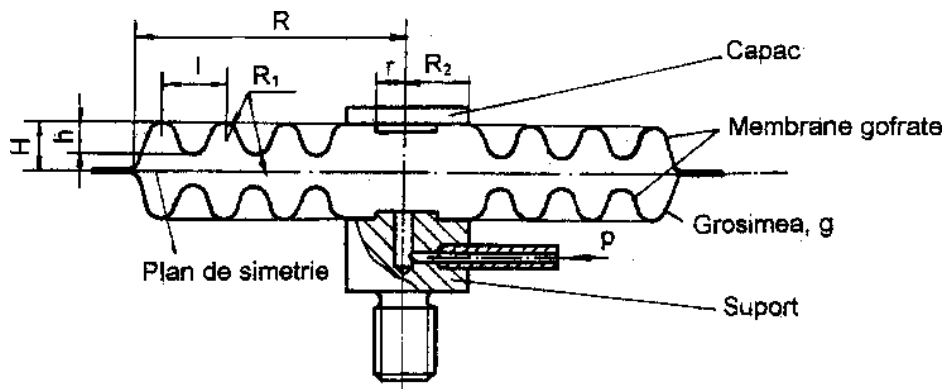
A. DEFINITION OF THE APPLICATION

A.1. Introduction

Mechanical systems sometimes contain two-dimensional elements with a much-reduced thickness, usually called membranes. Their structures during operation may be characterized by increased displacements with significant changes in load directions, usually statically applied (without considering time variation), or they may be dynamically stressed when the problem is to determine their response to certain loading regimes that vary in time. The problems of dynamic analysis of these structures using high-performance FEM-based programs utilize appropriate modules for resolution.

A.2. Description of the application

Elastic membranes, usually circular shape, can be components of precision manometers (altimeters, speedometers, vacuum gauges, etc.) with wide fluid pressure ranges. Also, elastic membranes can be used as pressure, force, or displacement transducers which, as a rule, are coupled with other transducers, such as piezoelectric or magnetoelastic ones. In the systems in which they are integrated, the supply fluid pressure has variable values. The dynamic response of the membrane structure to the action of the supply pressure has direct implications on the measurement accuracy of the output parameter.



A.3. Purpose of the application

This application aims to perform a linear dynamic analysis to determine the dynamic response of the corrugated elastic membrane structure within the manometric capsule of a dashboard instrument, shown in the figure above. The two corrugated membranes of the manometric capsule, made of stainless steel with a modulus of elasticity $E = 1,8 \cdot 10^5 \text{ N/mm}^2$ and a transverse contraction coefficient $\nu = 0,3$, have the dimensions $R = 48 \text{ mm}$, $r = 5 \text{ mm}$, $R_1 = 0,3 \text{ mm}$, $l = 10 \text{ mm}$, $R_2 = 10 \text{ mm}$, $h = 2 \text{ mm}$, $H = 2,5 \text{ mm}$.

The supply pressure has the variation presented in the figure below. To also take into account the subsystem driven by the membrane, represented in the figure above by a lid, a distributed mass is introduced on the flat surface, $m = 0,05 \text{ kg}$.

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.
- Associated masses.
- External loads of time-varying pressure type.
- Material characteristics.

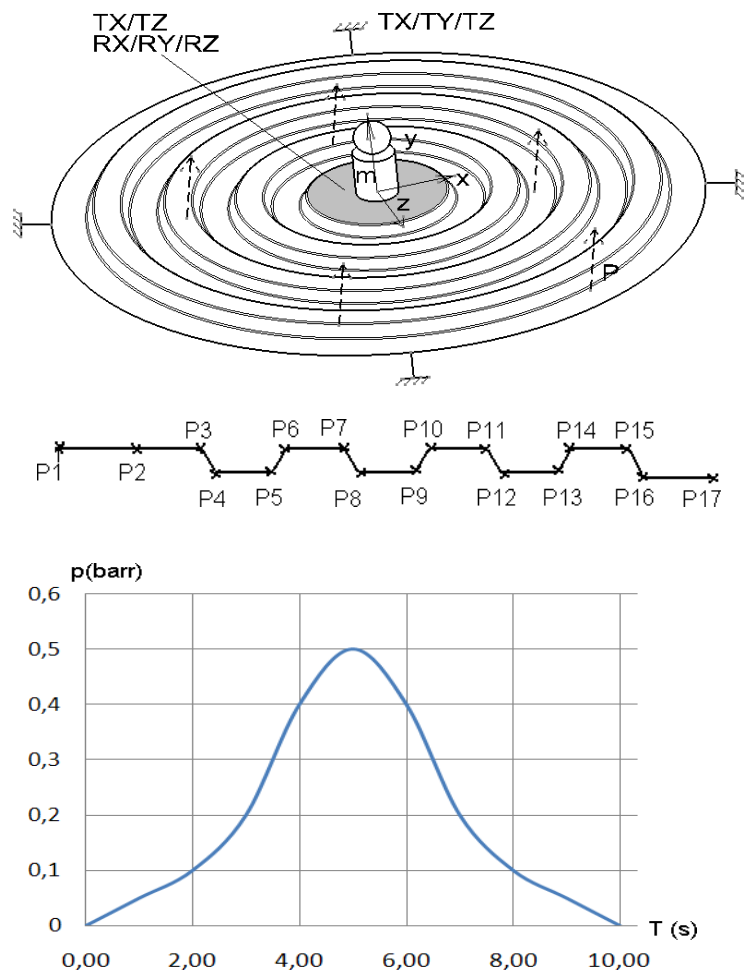
B.2. Description of the analysis model

The geometric shape and dimensions of the geometric model, from the first adjacent figures, are identical to those of the structure.

For the finite element analysis, in conditions as close as possible to the real model, the dynamic response is determined by canceling the translations of the points on the outer circumference and allowing displacement only along the OY axis for the points in the central zone.

The effect of the driven subsystems is taken into account by introducing the mass m at the center of the membrane.

This application aims to highlight the displacement and stress fields as well as their variation in time as an effect of loading with pressure p represented in the adjacent figure.



B.3. Establishing material characteristics

For the finite element analysis, the strength characteristics of the material (stainless steel) are:

- Longitudinal modulus of elasticity, $E = 180000 \text{ N/mm}^2$
- Transverse contraction coefficient (Poisson), $\nu = 0,3$

C. PRE-PROCESSING OF THE ANALYSIS MODEL

C.1. Geometric modeling

Activating the solid generation module and setting the unit of measurement for lengths

Start → Mechanical Design → Part Design → **New part**: New part name: Semicapsula.

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

Generating reference points

 **(Point)** → **Point Definition:** X 0mm, Y 0mm, Z 0mm; ↵OK. [Similarly, enter the coordinates of the other points (fig. B.2): P2(5,0,0), P3(10,0,0), P4(11,-2,0), P5(15,-2,0), P6(16,0,0), P7(20,0,0), P8(21,-2,0), P9(25,-2,0), P10(26,0,0), P11(30,0,0), P12(31,-2,0), P13(35,-2,0), P14(36,0,0), P15(40,0,0), P16(41,-2.5,0), P17(46,-2.5,0)].



Generating the polyline

 **(Polyline)** → **Polyline Definition:** [select with the mouse from the graphic area or from the tree structure points **Point.1** through **Point.17**, simultaneously generating their list in the Polyline Definition window], ↵ OK.

Observation

If the **(Polyline)** button is not found in the Toolbar area, execute the following sequence  **(Polyline)**

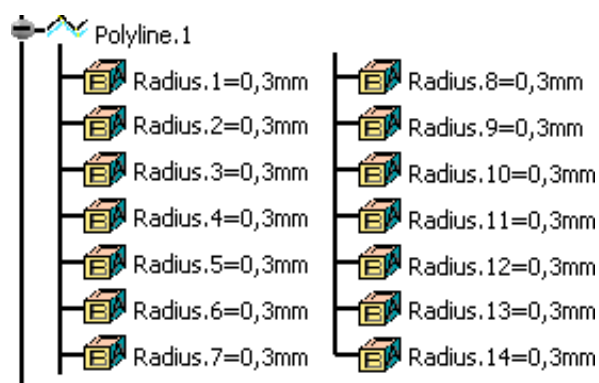
View → **Toolbars** → **Customize...** **Customize:** Commands, Categories ↓ **_All Commands,**
 Commands ↓ **Polyline...** → place the command symbol with the mouse (Drag&Drop) into
 the toolbar area, Close ↵ Close.

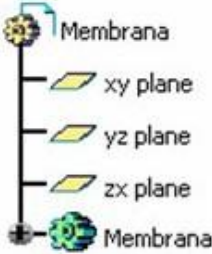
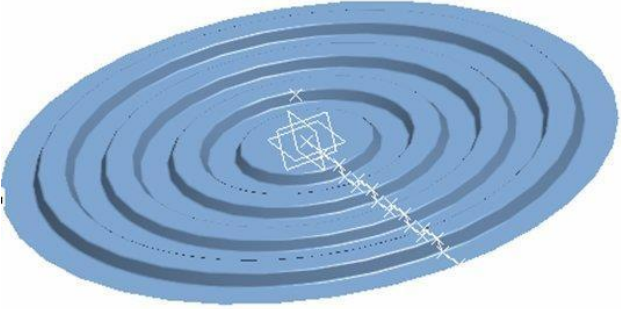
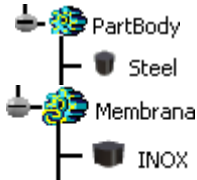
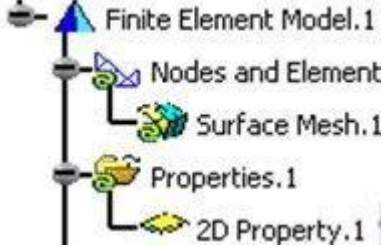
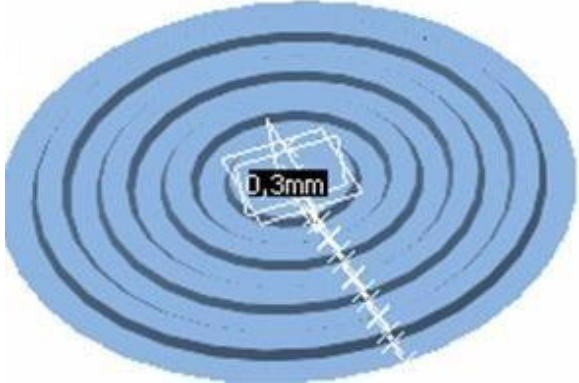


Generating the polyline with fillets

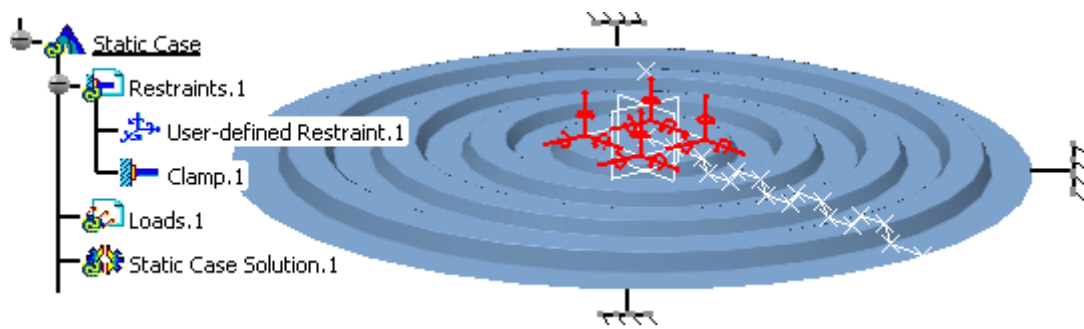
[Select the **Polyline.1** specification in the tree structure] → **Polyline Definition:** Point.3, Radius: 0,3 mm, ↵ OK.

[Repeat this sequence for the other points with fillets, **P4...P16**]



<i>Generating the membrane surface</i>	
 (Revolve) → Revolution Surface Definition : Angle 1: 360deg, Profile Selection [select Sketch.1 (or Polyline)], ↵OK.	
	
C.2. Modeling material characteristics	
[Select PartBody specification] →  (Apply Material) → Library (ReadOnly) : Metal, Steel, Apply Material; ↵OK. [Select Steel specification from the tree structure] → Properties : Feature Properties, Feature Name: INOX; Analysis, Young Modulus 180000 N/mm ² , Poisson Ratio 0,3; Apply; ↵OK.	
C.3. Finite element modeling	
<i>Activating the advanced finite element meshing module</i>	
Start → <u>A</u> nalysis & Simulation → <u>A</u> dvanced Meshing Tools → New Analysis Case : Static Analysis ↵OK.	
<i>Selecting the finite element type and meshing characteristics</i>	
 (Surface Mesher) → [select the surface, Membrana] → Global Parameters : Shape, Type: Linear, Mesh size: 1 mm, ↵OK.	
<i>Activating the finite element analysis module</i>	
Start → <u>A</u> nalysis & Simulation → Generative Structural Analysis.	
<i>Introducing material properties</i>	
 (2D Property) → 2D Property : Supports: [select from the tree structure the specification, Surface Mesh.1] → Thickness: 0,3 mm ↵OK.	
	
C.4. Modeling geometric constraints	
<i>Introducing displacement restrictions</i>	
 (User-defined Restraint) → Restraints → Supports: 1 Face [select the central surface of the model], ☒ Restrain Translation 2, ↵OK.  (Clamp) → Clamp : Supports [select the outer circle associated with the model surface],	

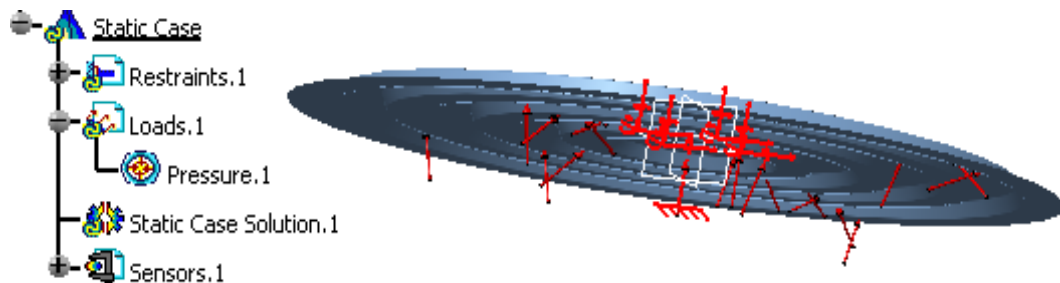
↓OK.



C.5. Modeling loads

Introducing maximum pressure

 (Pressure) → **Pressure** → Supports: [select the inner surface of the model] → Pressure: $0,5 \cdot 10^5 \text{ N/m}^2$, ↓OK.



D. SOLVING THE FINITE ELEMENT MODEL

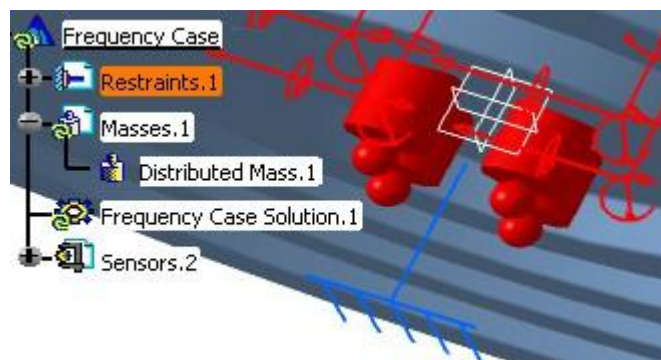
D.1. Launching the calculation module

 (Compute) → **Compute**: ↓All, ↓OK → **Computation Resources Estimation**, ↓Yes → **Computation Status**.

D.2. Activating the natural frequency analysis module

Insert → Frequency Case → **Frequency Case**: ☒ Restraint, ☒ Reference: Restraints.1 ☒ Masses, ☒ New, ☐ Static Case Solution, ☒ Hide existing analysis case, ↓OK.

 (Masses) → **Distributed Mass** Name: Distributed Mass.1, Supports: [select the central surface of the model], Mass: 0,05 kg, ↓OK.



D.3. Re-launching the calculation module

 (Compute) → **Compute**: ↓All, ↓OK → **Computation Resources Estimation**, ↓Yes → **Computation Status**.

D.4. Activating the dynamic analysis module

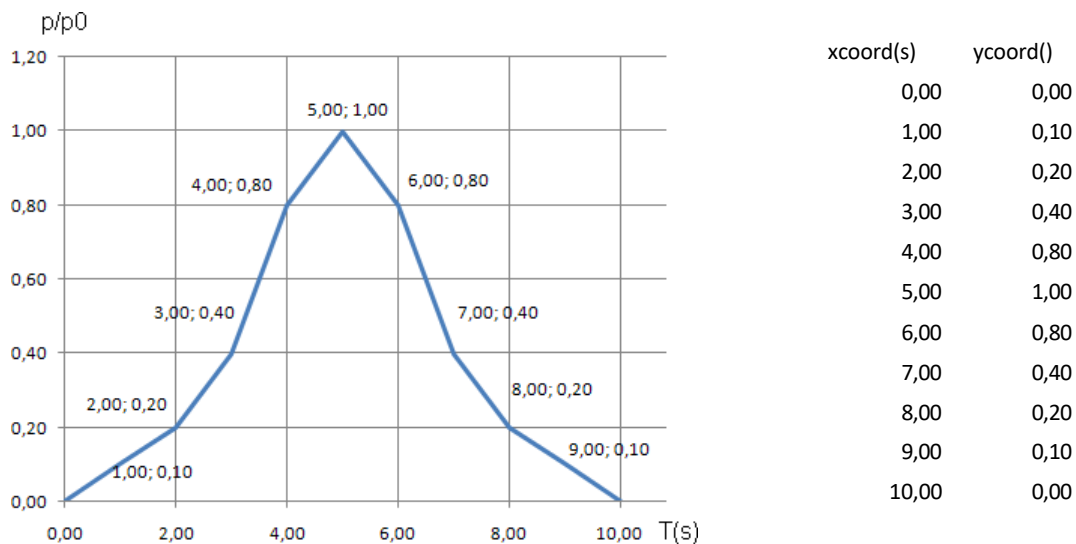
Insert → Transient Dynamic Response Case → **Transient Dynamic Response Case**:

Transient Case Solution, Reference: [select from the tree structure the specification, **Frequency Case Solution.1**], Load excitation, ☒ New, ☐ Restraint excitation, ☐ Hide existing analysis case, ↵OK.

D.4. Modeling the variable pressure loading

Defining the modulation

[Generate in Excel (or Notepad) the table below associated with the points on the approximation graph in the figure below; Save the file under the name **Book1tmp.xls**].



Introducing variable pressure



(Time Modulation) → **Time Modulation**: Name: Time Modulation.1, Browse: *Book1tmp.xls* [select the previously generated file], ↵OK.

Activating the variable pressure load

[Select with the mouse (double click) in the tree structure] (Load excitation.1) → **Load Excitation Set**: Name: Load excitation.1, Select Load: Loads.1, Select Modulation: TimeModulation.1, Selected factor:1, ↵OK.

D.5. Re-launching the calculation module



(Compute) → **Compute**: ↵All, ↵OK → **Computation Resources Estimation**, ↵Yes → **Computation Status**.

E. POST-PROCESSING OF RESULTS

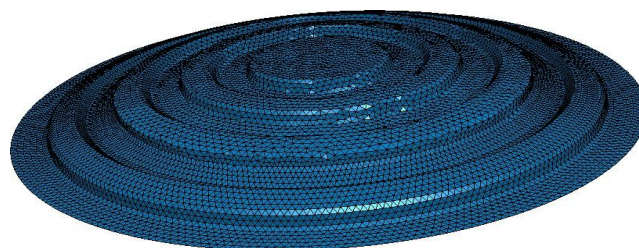
E.1. Visualizing the deformed state/Animation



(Deformation).



(Animate) → **Animate**: Steps number: 20 (other parameters can also be modified), ↵Close.



Observation

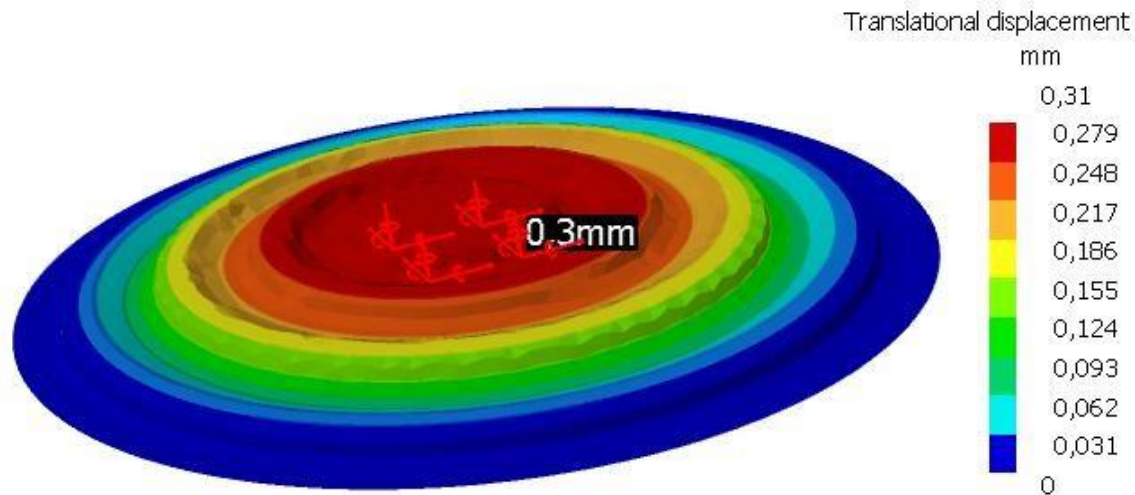
To modify the amplitude of displacements during animation for better visualization, follow the sequence



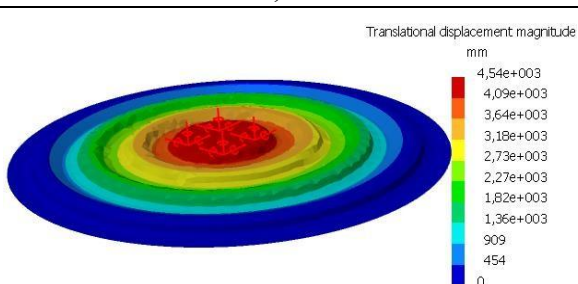
→ **Amplification magnitude:** Factor: 1 (other parameters can be set), **OK**.

E.2. Visualizing the displacement field (static analysis)

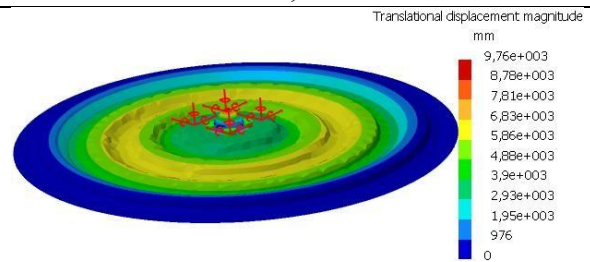
(Displacement).

**E.3 Visualizing displacement fields (natural frequency analysis)**

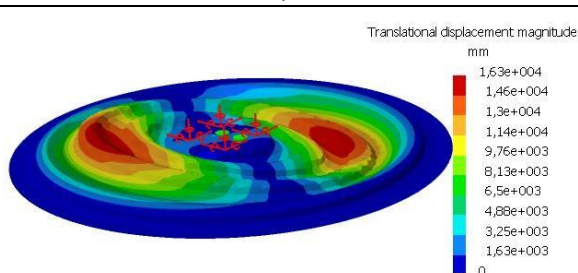
409,24 Hz



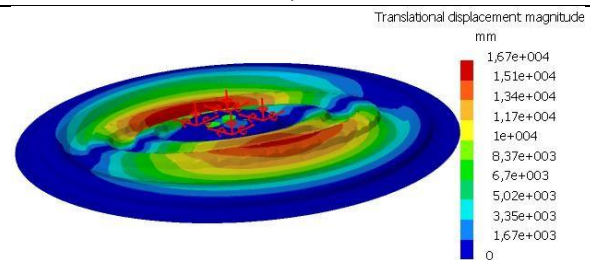
2419,26 Hz



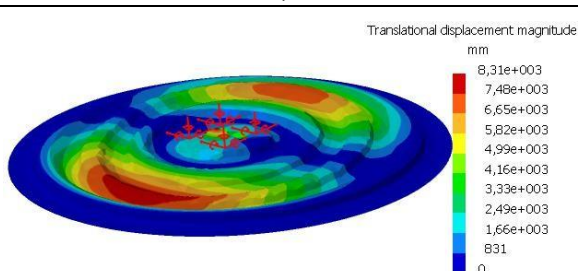
2720,54 Hz



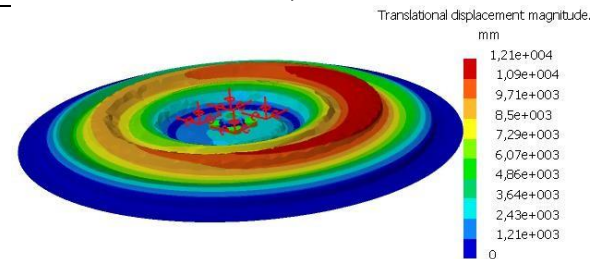
2738,46 Hz

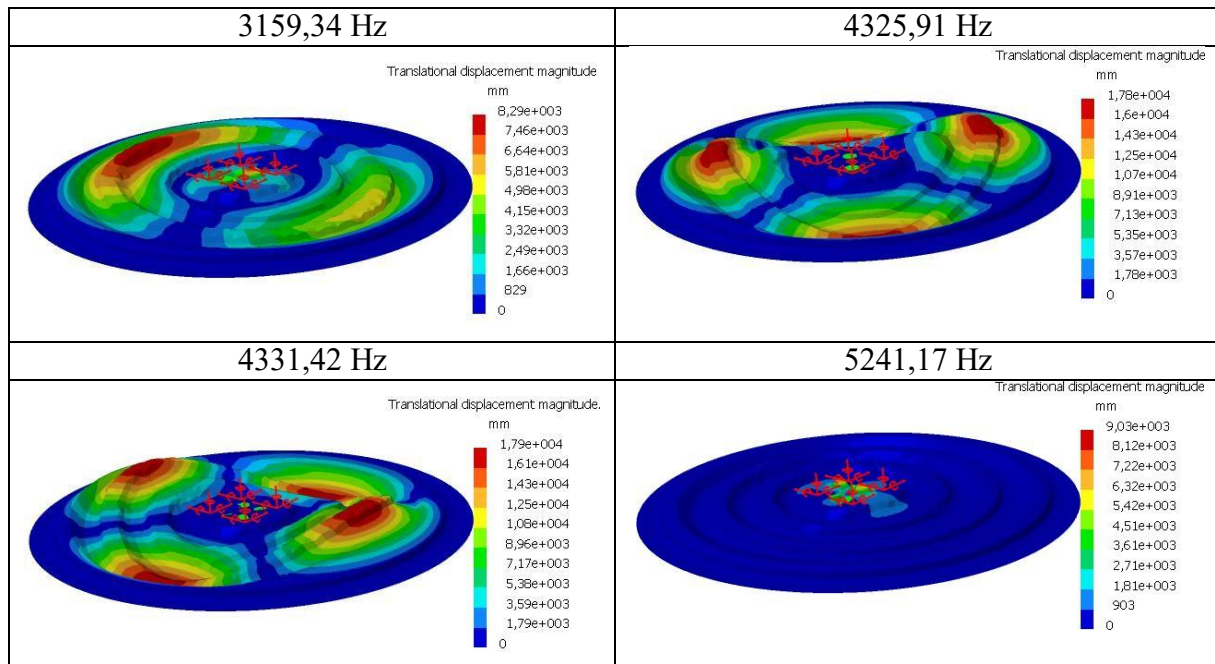


3083,45 Hz



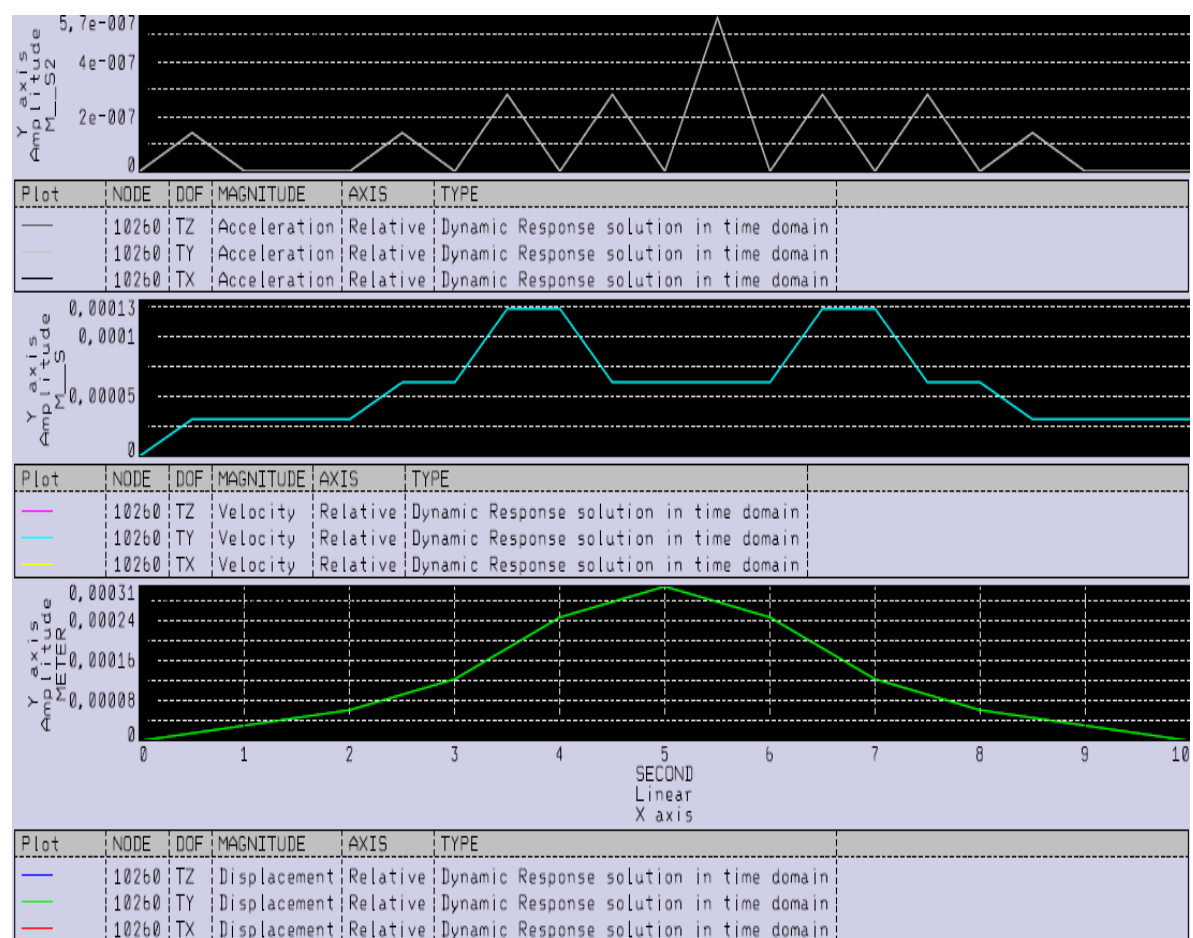
3097,46 Hz





E.4. Visualizing the dynamic response associated with displacements (dynamic analysis)

[Select with the mouse from the tree (right click) **Transient Dynamic Response Solution.1**] → Generate 2D Display → **New Function Display:** Next → 3 graphs → Finish → **Selected Data** → Node: [select a node from the central area of the membrane surface], OK.



F. CONCLUSIONS

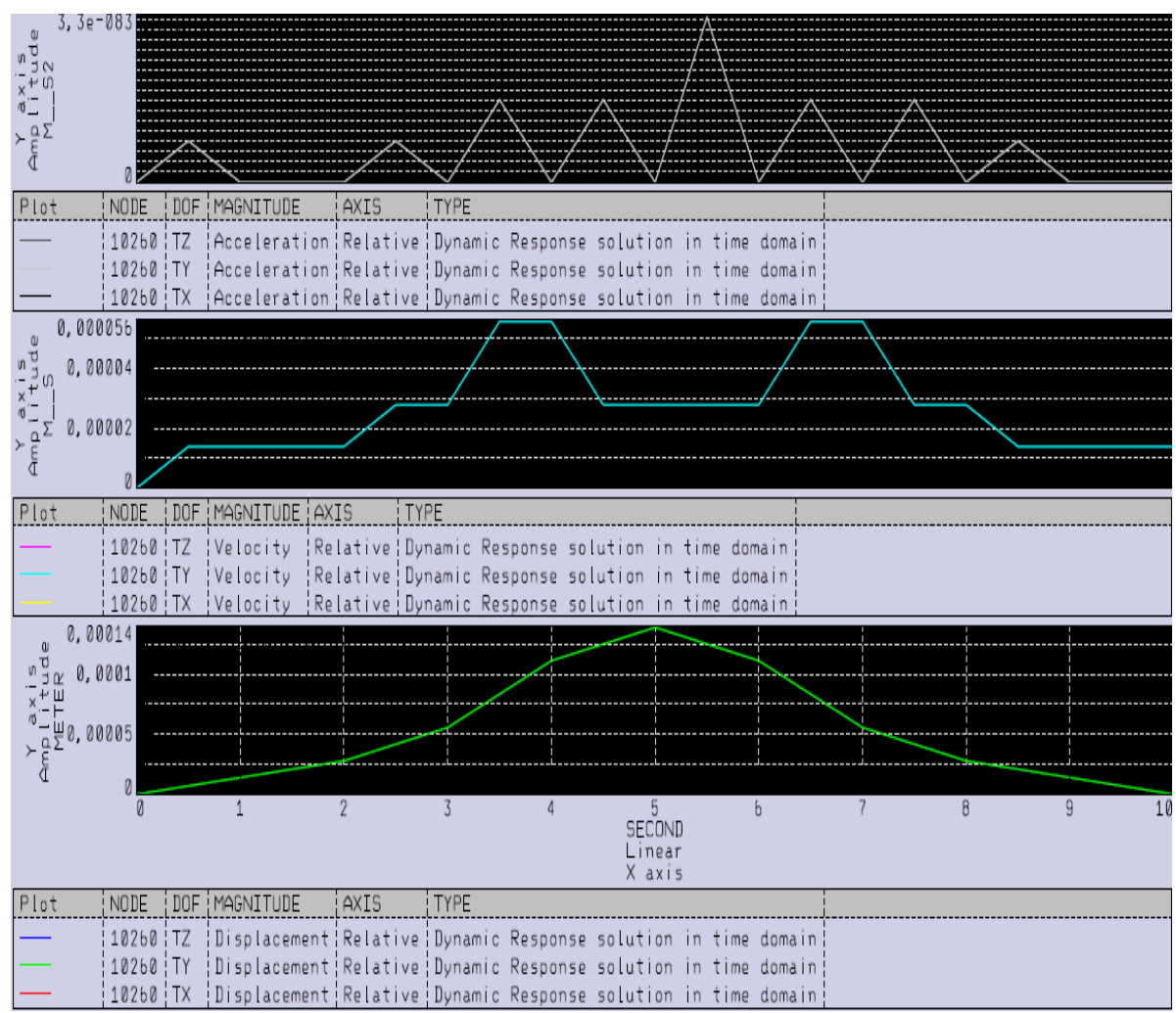
In order to determine the dynamic response of the membrane structure, static, natural frequency and dynamic finite element analyses were carried out successively. Figures E.1 and E.2 show the deformations and displacements for the static analysis. The visualization of the natural vibration modes associated with the natural frequencies is presented in Fig. E.3.

The dynamic response can be followed in the diagrams in Fig. E.4 and below, which highlight the variations in the displacement, velocity and acceleration of a real point in the central part of the membrane, in relation to time.

When taking into account the damping generated by the frictions in the subsystem driven by the membrane, according to the sequence below, a reduction in the amplitudes, displacements, velocities and accelerations is observed. For a damping factor of 10%, the maximum displacement of the central point is reduced from 0.31 mm to 0.14 mm.

[Select with the mouse in the tree view, (double click) Damping.1] → **Damping Choice:**

Name: Damping.1, Damping type: Modal damping,  (Component edition) → Damping Definition, Global ratio: 10%, ↵OK, → ↵OK.



G. EXERCISE

Determine the dynamic response of the DIN 2093-disc spring, with dimensions: $D = 40$ mm, $d = 16.3$ mm, $g = 2$ mm, $h = 8$ mm, shown in the figure below. It is made of

steel 50VCr11A, hardened to 50 ... 55 HRC and is loaded by a uniformly distributed axial force on the inner edge of 800 N. The friction between the inner edge and the seating surface will be taken into account by a damping factor of 15%.

