Application: AEF-A.13

Non-metallic elastic elements

KEY WORDS
Transient structural analysis, Nonlinear material, Hyperelastic material, 3D geometric model, 3D finite element, Damping elements, Own vibrations, Variable load

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

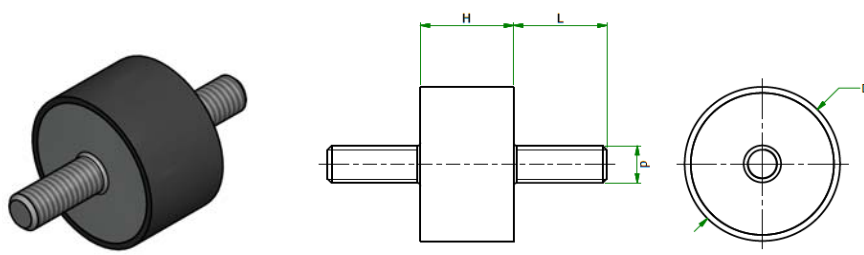
Many technical products contain mechanical elements that have distinct compact structures, required by the main function to be performed. Representative of this group of components are the elastic elements (springs), the damping elements, the housing support elements, etc. The specificity of these elements, as a rule, is given by their fixed or quasi-fixed connections with the neighboring parts.

The finite element analysis of these components, in order to obtain precise results, presupposes the accurate definition of the solid model, of the restrictions imposed by the connections with the neighboring elements, as well as of the loads. This application will aim to perform a transient dynamic analysis to determine the response of the target structure to tasks that vary over time. If the effects of inertia and damping are not significant, another analysis can be performed - static structural with variable force.

A.2 Application description

Elastic elements made by rubber are frequently used in the construction of mechanical systems due to their elastic capacity and especially their damping capacity and, in many cases, due to their lower cost.

The buffer in the adjacent figure is composed of a cylindrical piece of rubber to which are attached two flat metal reinforcements.



These elements are frequently introduced in the subsystems of motor vehicles having the role of supporting parts or as elastic quasi-couplings with damping, bringing the following advantages: they eliminate wear and noise, dampen vibrations, have moderate costs and unpretentious maintenance. On the other hand, these elements have a shorter service life than steel due to the decrease in the strength and elasticity properties of rubber over time (aging process).

A.3 Application goal

The aim of this paper is to determine the displacement, deformation and stress fields of the standardized rubber buffer structure, type 497719, usually used to support the muffler in some vehicles. The rubber pad has the following dimensions: D = 70 mm, threaded rods M10, L = 43 mm, H = 70 mm. The loading and fixing of the studied element is done by means of the M10 metal rods integral with the flat metal reinforcements. The metal reinforcements are considered to be rigid and non-deformable in relation to the rubber mass of the element.

B. PREPARATION OF THE MODEL FOR ANALYSIS

B.1 The model definition

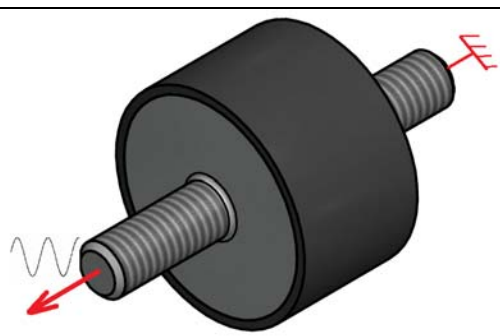
In order to draw up the finite element analysis model associated with the above application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The geometric shape and dimensions of the rubber pad are shown in the adjacent figure. For the analysis, the structure of the buffer is modeled with 3D finite elements and, therefore, the geometric model is identical to the solid model.

In order to simulate the behavior of the buffer as close to reality as possible, considering its loading by means of an M10 threaded rod and of the reinforcements made of steel, characterized by increased rigidities, the fixing constraints and loads will be introduced directly on the faces of the rubber cylinder. In order to simulate the connection with the outside by means of the external reinforcement, boundary conditions are introduced which imply translation restrictions after the three directions of the XYZ coordinate system for all points of the surface.



ITEM	D	H	d	L
497719	70	70	M10	43

B.3 Characteristics of the material
The analyzed element is made of neoprene rubber, with a hardness of 70 Sh and the following deformation constants: A_{10} = 0.177 N / mm2, A_{01} = 0.045 N / mm^2 and D1 = 333 N / mm^2. These characteristics, in the area of small deformations, correspond to the following physical parameters of analysis: • modulus of longitudinal elasticity, E = 400 MPa; • transverse contraction coefficient (Poisson), ν = 0.49.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Creating of the project

The following commands will be executed, in the order shown:

Λ, Toolbox: ↵ ⊟ Analysis Systems → ↵↵ Transient Structural (the subproject window appears automatically) → Save As... → File name: *Rubber element* → Save.

Problem type setting (3D)

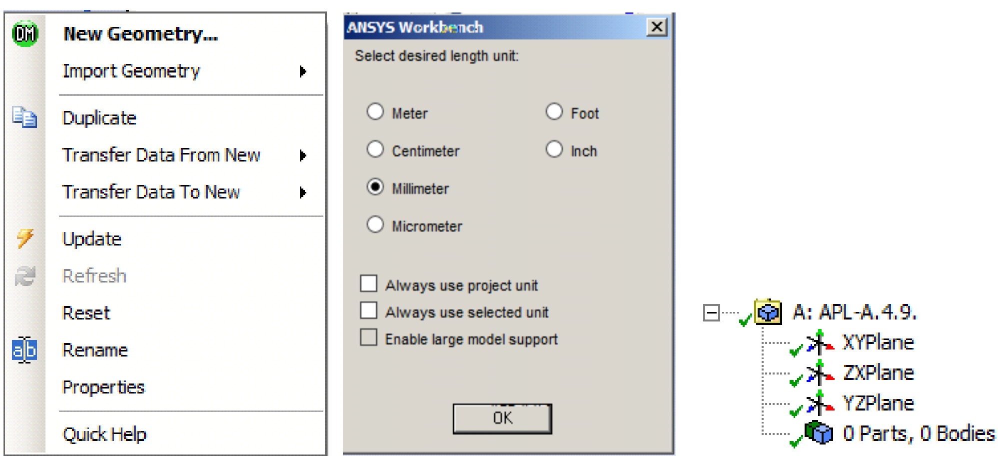
A: ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from list ↵▾, ↵ 3D] → [close window ↵X].

Save of the project

↵ Save As... → Λ Save As, File name: [input name, *Rubber element*] → ↵ Save.

Setting the unit of measure for lengths

Λ, Toolbox: ↵ ⊟ Analysis Systems → **ANSYS Workbench**: Select desired length unit: ◉ Millimeter → ↵ (OK) → DM → Generate.



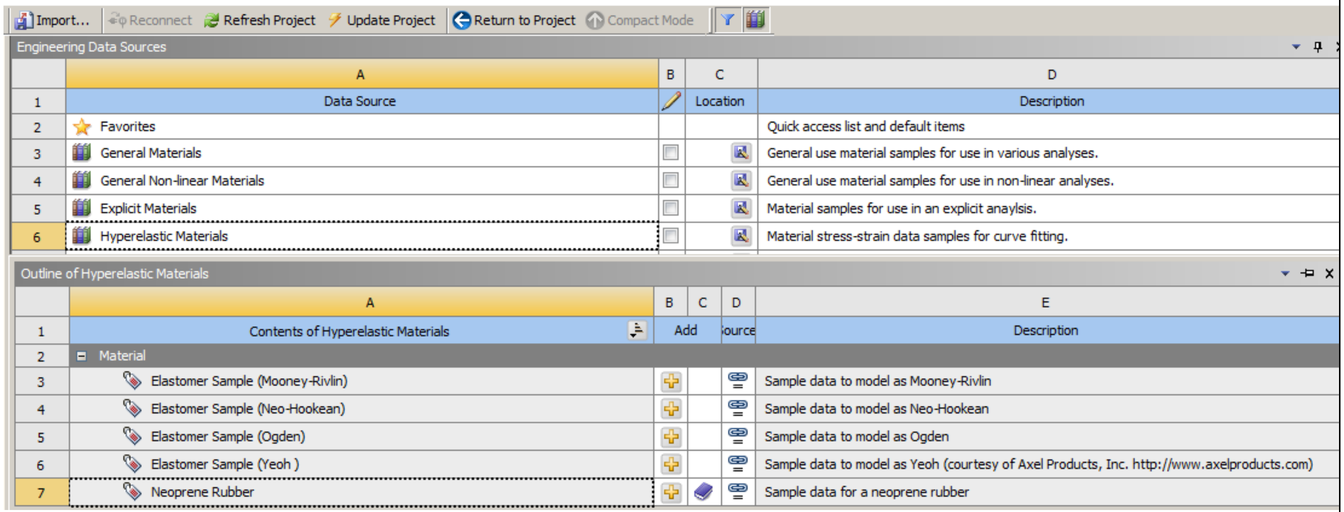
C.2 Modelling of material characteristics

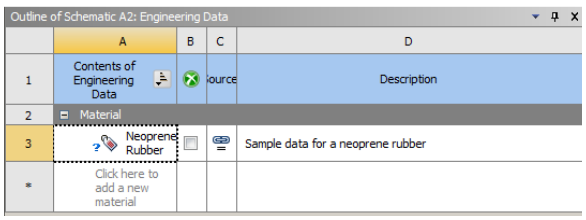
Choosing of hyperelastic material (neoprene rubber):

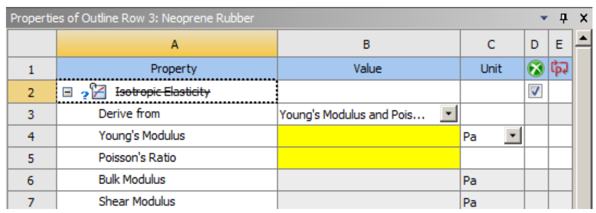
Λ → ↳ Engineering Data → Edit... → Properties of Outline Row 3: Structural Steel (by default, the program opens the *Structural Steel* material, to be changed to *Neoprene Rubber*, chosen from the material database).

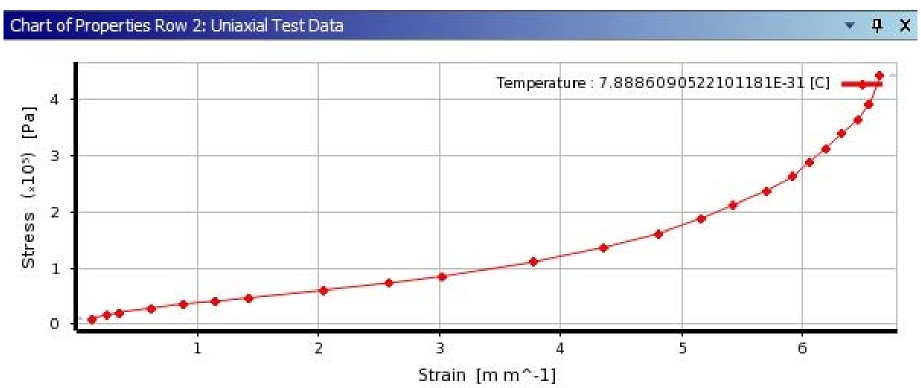
Λ → Outline of Schematic A2: Engineering Data → ↳ Structural Steel → Delete → ↵ → Engineering Data Sources → ↵ Hyperelastic Materials → Outline of Hyperelastic Materials → ↵ Neoprene Rubber → ↵ (Add to A2: Engineering Data) → Properties of Outline Row 7: Neoprene Rubber → the material characteristics stored in the program database are accepted → Update Project → Return to Project, except for a few properties: density (the value of

1000 kg / m^3 will be entered) as well as the modulus of longitudinal elasticity (E = 400 MPa) and the Poisson transverse contraction coefficient (ν = 0.49).









C.3 *Creating the geometric model*

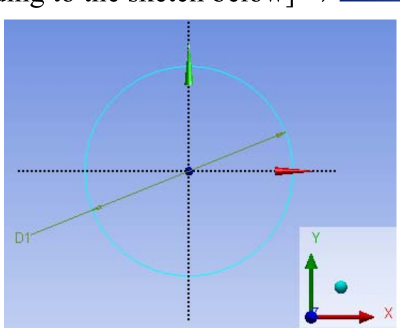
C.3.1 *Uploading DesignModeler Module (DM)*

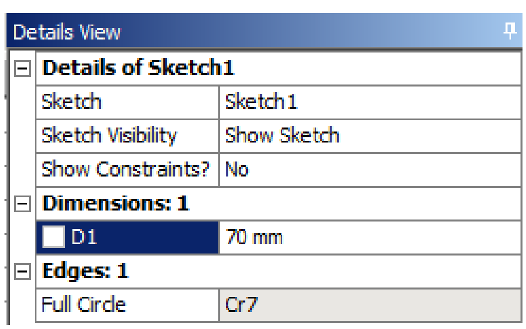
, Project Schematic: ↳ Geometry → ↵ **New Geometry...** → ANSYS Workbench: ↵ Millimeter, ↵OK.

C.3.2 *Creating the rubber component*

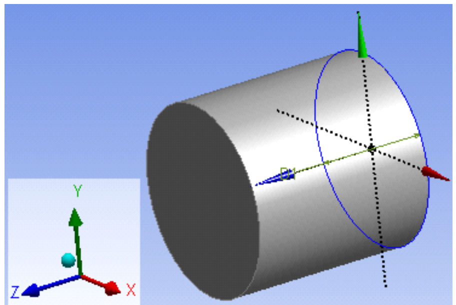
<u>*Creating the body*</u>

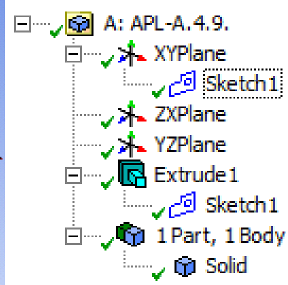
→ Modeling → ↵ ✓ XYPlane → Sketching → → <u>Draw</u> → Circle [In the 2D modeling area a circle is created according to the sketch below] → Details View → **Dimensions: 1** → D1 = 70 → Generate.





DM → Modeling → Extrude → Details View → **Details of Extrude 1** → Geometry: Sketch1 → Depth: 70 → Generate





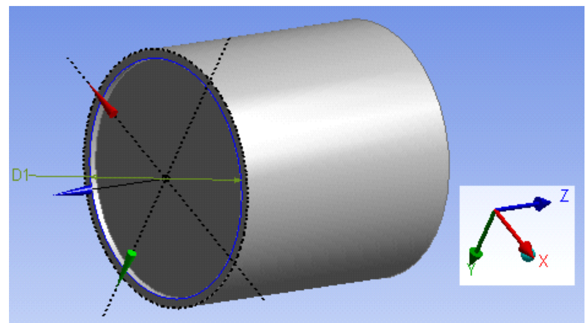
Making clearances at the ends

A 65 mm diameter and 2 mm deep recesses shall be drilled to simulate the position of the metal reinforcement of the buffer.

DM → Modeling → (New Plane) → Details View → **Details of Plane 1** → Type: From Plane → Base Plane: XY Plane → Generate → Sketching → → Plane 1 → Circle [In the plane P1 draw a concentric circle with circle D1 and diameter D2 = 65 mm] → Details View → **Dimensions: 1** → D1 = 65 → Generate.

DM → Modeling → Extrude → Details View → **Details of Extrude 2** → Geometry: Sketch2 → Operation: Cut Material → Depth: 2 → Generate

Repeat the above steps to clear the other end of the cylinder.



C.3.3 Saving the geometric model

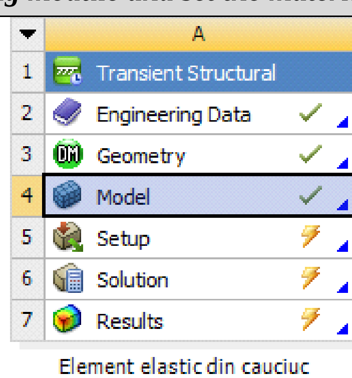
DM → (Save Project) → File → Close Design Modeler.

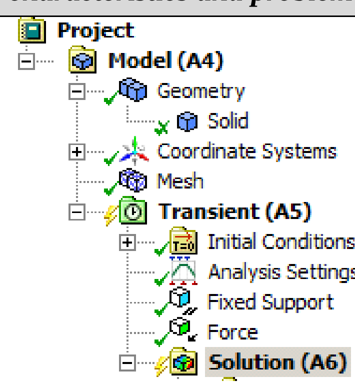
C.4 Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

Λ → ↵↵ Model → launching the module *Mechanical [ANSYS Multiphysics]*.

M, Outline: ↵ Solid → Details of "Solid" → **Material** → Assignment: Neoprene Rubber.

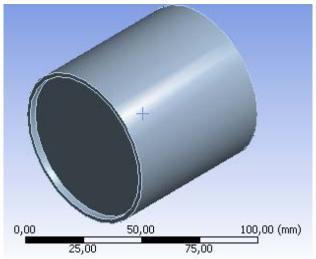




C.4.2 Model discretization and finite element size setting

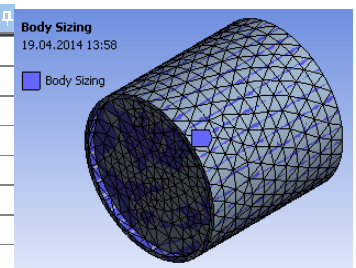
M, Outline: ↵ Mesh → Mesh Control → Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ 3D body geometry, using the selection filter (Body)] Apply; **Definition**

Element → Size: Default → Nonliniar Effects: Yes → Update. For a proper view of the discretization, select: ↵ Show Mesh.



Details of "Body Sizing" - Sizing

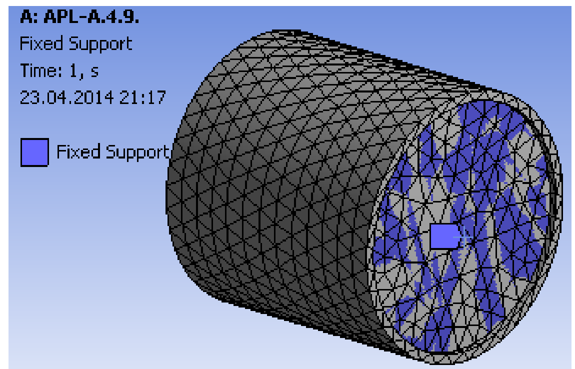
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	Default
Behavior	Soft



C.5 Supports and restraints modelling

Input restraint

M, Outline: ↵ Static Structural (A5) → Supports → Fixed Support → Details of "Fixed Support" → **Scope** → Geometry: [the surface at the end of the spring at a height of 2 mm shall be selected with ↵ using the selection filter (Face)] → Apply.



C.6 Load modeling

Input forces

Because the part being used is used to secure a car's exhaust pipe, the stresses will not only be static, but will vary over time, depending on the vibration of the drum.

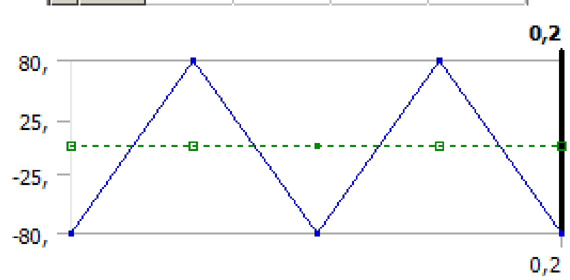
A sinusoidal, time-varying force-type load will be used in this study. Upload values will need to be entered in tabular form.

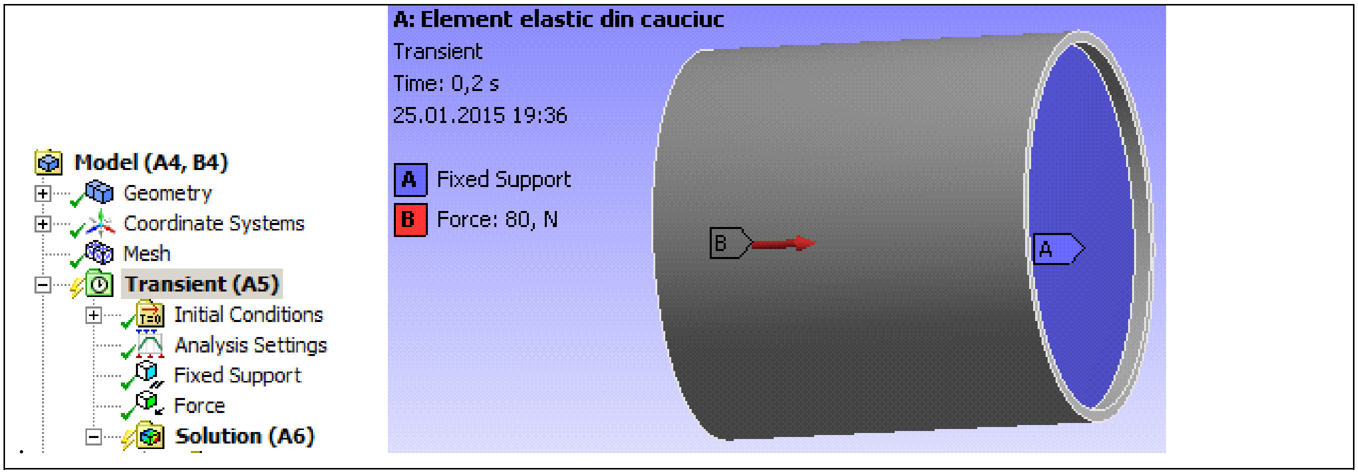
M, Outline: ↵ Analysis Settings → Details of "Analysis Settings" → **Step Controls** → Number of Steps: 1, Current Step Number: 1, Step End Time: 0,5 s, Auto Time Stepping: Off.

M, Outline: ↵ Transient (A5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [selecting with ↵ the surface from the end of the buffer at distance 68 mm, using option (Face)] → Apply; **Definition** → Define by: Components; X Component = 0 N, Y Component = 0 N, Z Component: Tabular Data → Tabular Data → Tabular Data (the table with the values of the loads presented next will be completed). The constraints and loads of the resort will look like the figure below

Tabular Data

	Steps	Time [s]	X [N]	Y [N]	Z [N]
1	1	0,	= 0,	= 0,	-80,
2	1	5,e-002	= 0,	= 0,	80,
3	1	0,1	0,	0,	-80,
4	1	0,15	= 0,	= 0,	80,
5	1	0,2	= 0,	= 0,	-80
*					





D. SOLVING THE FEA MODEL

D.1 Setting results

In order to select the final data types to be analyzed after the launch of the calculation module, follow the series of commands presented below.

Total deformation setting

M, Outline: ↳ Solution (A6) → ↲ Insert → ↲ Deformation → ↲ Total.

Equivalent stress setting

↳ Solution (A6) → ↲ Insert → ↲ Stress → ↲ Equivalent (von-Mises).

One direction deformation settings

↳ Solution (A6) → Deformation → Directional.

Next, set the other types of results to be analyzed:

↲ Solution (A6) → Stress → Error.

↲ Solution (A6) → Strain → Equivalent (von-Mises).

↲ Solution (A6) → Energy → Strain Energy.

D.2 Launching the solving module

M, Outline: ↲ Analysis Settings → Details of "Analysis Settings" → **Solver Controls** → Large Deflection : On

↲ Solution (A6) → Solve.

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu bars:

Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

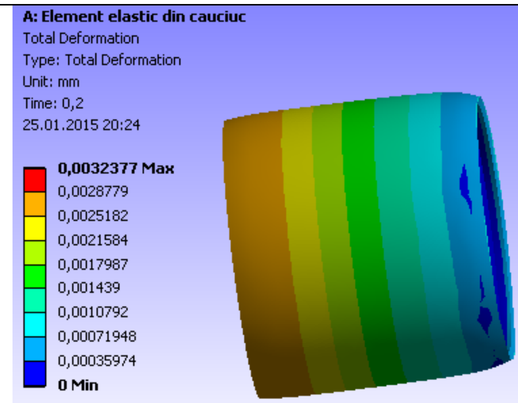
The section will be used to view the analyzed part in section (*New Section Plane*) located on the Desktop and a section plan will be chosen.

Total deformation view

↲ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation.
The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command.

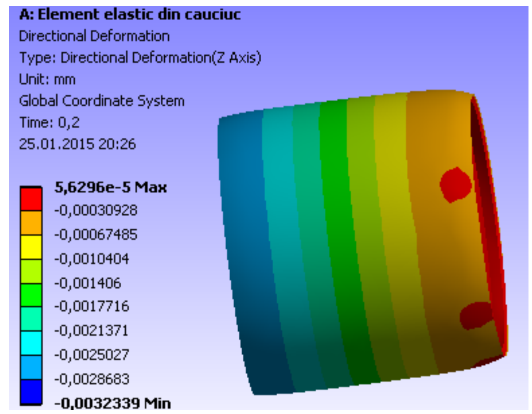


Visualization of the deformation in a certain direction

↵ Solution (A6) → Directional Deformation → Graph → Animation.

If you want to view in another direction, follow the steps below:

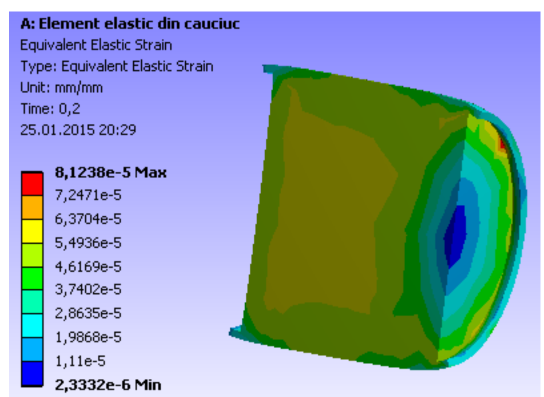
↵ Solution (A6) → Directional Deformation → Details of "Directional Deformation" → **Definition** → Orientation : Z Axis → Solve.



E.2 Visualization of equivalent stress fields, structural error and acceleration

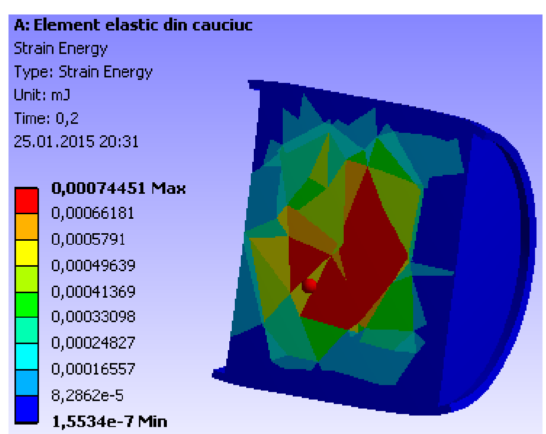
Equivalent Elastic Strain

↵ Solution (A6) → Equivalent Elastic Strain → Graph → Animation.



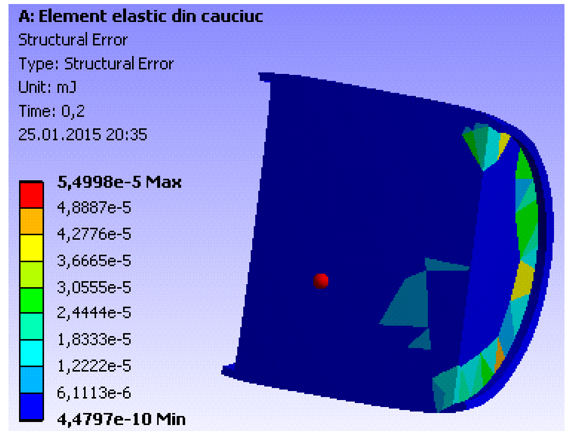
Strain Energy

↵ Solution (A6) → Strain Energy → Graph → Animation.



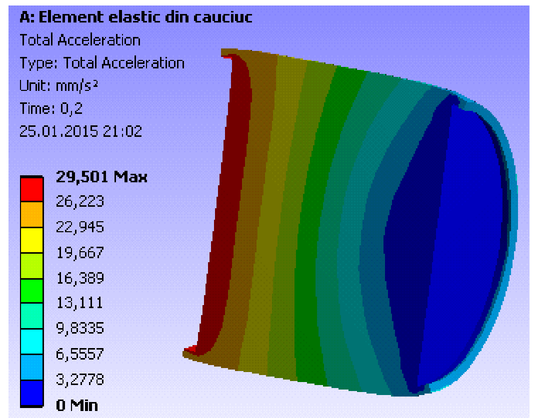
Structural Error

↵ Solution (A6) → Structural Error → Graph → Animation (sau Tabular Data).



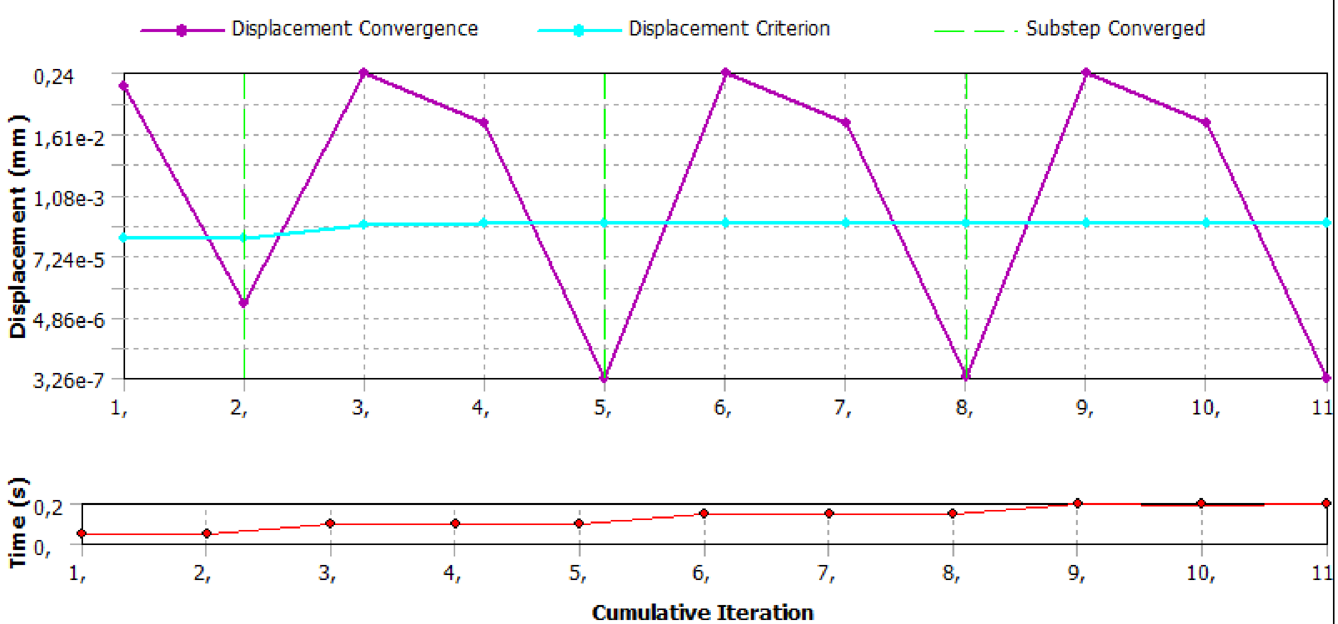
Total acceleration

↵ Solution (A6) → Total Acceleration → Graph → Animation (sau Tabular Data).



E.3. Visualizing the convergence of solutions

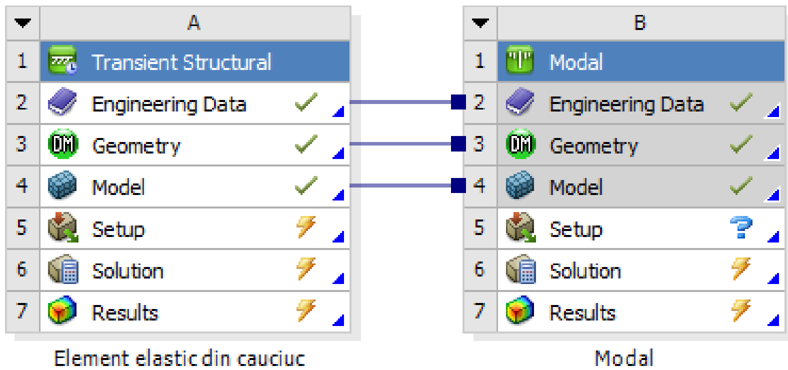
↵ Solution (A6) → Solution Information → Details of "Solution Information" → **Solution Informations** → Solution Output: Displacements Convergence.



F. ANALYSIS OF RESULTS

In order to avoid loading the buffer with a variable load that overlaps with its own vibration values, it is recommended to perform a modal analysis beforehand to determine them. For the rubber buffer, the modal analysis is performed as follows.

→ Project Schematic → holding down ↵ on the command in the analysis structure ⊟ Analysis Systems pulls on the command Model ✓ from structure of *Transient Structural,* until it turns into Share B3, when the mouse key is released → OK



The natural vibration frequencies obtained by the modal analysis have very low values, far from the values of the stress frequencies induced by the operation of an internal combustion engine. From the point of view of deformations, it is observed that, following a variable stress in the range (-80, +80) N, they are in a very small value range, of the order of 0.005 mm, which does not cause a rapid deterioration. of the tampon.

	Mode	Frequency [Hz]
1	1,	0,
2	2,	0,
3	3,	0,
4	4,	1,8714e-004
5	5,	1,2895e-003
6	6,	1,6363e-003

G. CONCLUSIONS

Modeling and analysis with finite elements in this paper were made especially for teaching purposes following the user's initiation with the main stages of developing an FEA application in ANSYS Workbench, which emphasizes, in particular, the modeling and analysis of an elastic element made of -a material with nonlinear behavior (hyperelastic material - neoprene rubber).

The analysis algorithm for the Transient Structural type was highlighted, introducing time-varying stresses. At the same time, the importance of performing a modal analysis was highlighted in order to identify the values of the own vibrations, in order to be used in the design activity.