



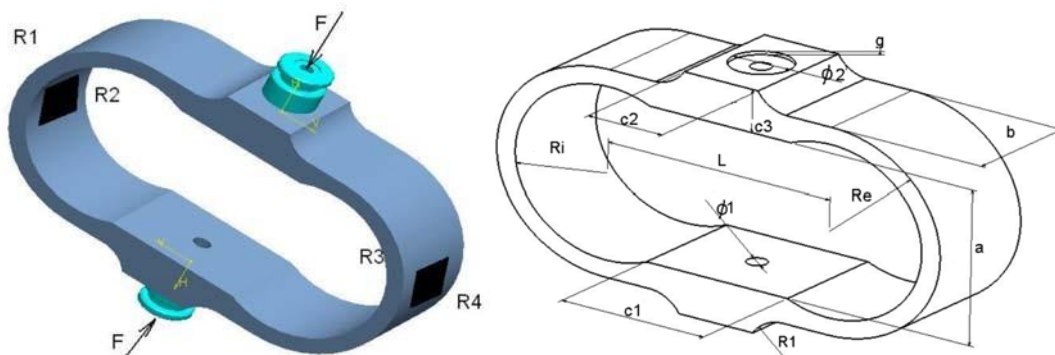
LINEAR STATIC ANALYSIS OF THE TENSOMETRIC CAPTOR

INTRODUCTION

It is comprised of many technical products, mechanical components that have distinct compact structures required by the main function to be fulfilled. Representative of this group of components are elastic elements (springs), damping elements, supporting elements (housings), etc. The specificity of these elements, as a rule, is given by their fixed or quasi-fixed links with adjacent parts. The finite element analysis of these components for accurate results requires accurate definition of the solid model, the constraints imposed by the links with the adjacent elements, as well as the loads.

DESCRIPTION OF THE APPLICATION

In the test stands, as well as devices or products (balances, process automation systems, etc.), there are subsystems for the measurement of forces based on transducers that have attached elastic elements with suitable sensors. Frequently, for the measurement of forces, electro-resistive tensiometric transducers (tensiometric marks) consisting of a thin conducting wire, bonded to a paper support or insulating material, are used. The transducers glued to an elastic element (captor) are deformed due to the action of the external force and the consequence of changing the length and cross-section of the wire cause changes in the electrical parameters (intensity and voltage) in a suitable circuit. The elastic force trap F shown in the figure below is support for the tensiometric brands R1, R2, R3, R4 usually connected in a Wheatstone bridge circuit. As a consequence of the displacements of the points under the tensiometric support, variations in the lengths (sections) of the conductive threads and, implicitly, variations in the electrical resistances occur.



PURPOSE OF THE APPLICATION

In this case, the analysis of the fields of displacements, deformations and stresses in the elastic cap structure is presented. The analysis was carried out in order to establish the areas with maximum deformations for the application of the tensiometric marks and, on the other hand, the non-determination of the flow limit of the material under the conditions of the maximum load. The presented cap with the following dimensions: $R_e = 60$ mm, $R_i = 50$ mm, $R_1 = 20$ mm, $L = 120$ mm, $a = 90$ mm, $b = 60$ mm, $c_1 = 76$ mm, $c_2 = 45$ mm, $c_3 =$ mm, $g = 2$ mm, $\phi_1 = 10$ mm, $\phi_2 = 30$ mm, is designed to operate in the force range of 5-10 kN, applied to it by direct contact on the front surfaces of the lemons with the load parts adjacent. The material of the elastic element is steel, OLC 45, improved to 250-270 HB.

FEA Application A.2.4

LINEAR STATIC ANALYSIS OF TENSOMETRIC CAPTOR TYPE ELEMENTS

A. DEFINING THE APPLICATION

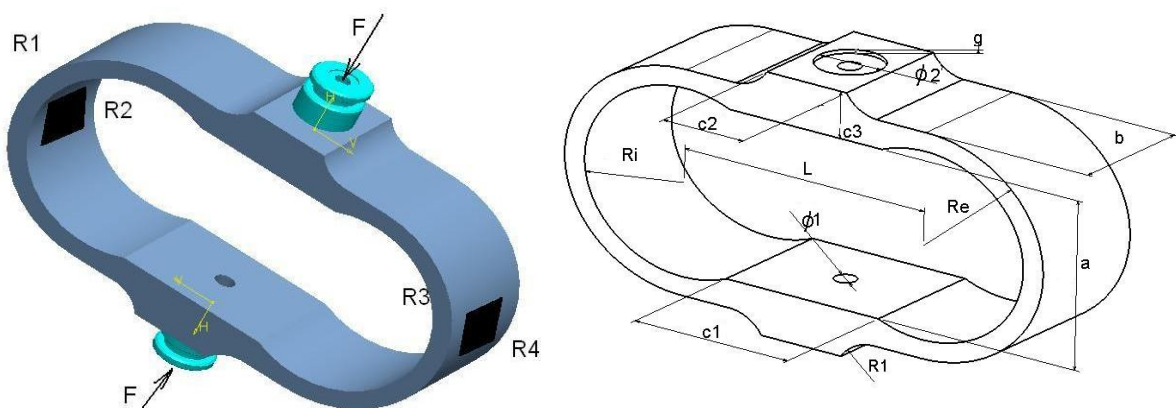
A.1. Introduction

Many technical products contain mechanical components that have distinct compact structures required by the main function to be fulfilled. Representative of this group of components are elastic elements (springs), damping elements, and support elements (housings), etc. The specificity of these elements, as a rule, is given by their fixed or quasi-fixed connections with adjacent parts. The finite element analysis of these components, to obtain precise results, requires the accurate definition of the solid model, the constraints imposed by the connections with neighboring elements, as well as the loads.

A.2. Description of the Application

Test stands, as well as some devices or products (scales, process automation systems, etc.), include subsystems for measuring forces based on transducers that have elastic elements with appropriate sensors attached. Frequently, electro-resistive tensiometric transducers (strain gauges) are used for measuring forces. These consist of a thin conductive wire bonded to a paper or insulating material support. The transducers, when glued onto an elastic element (captor), deform due to the action of the external force. As a consequence of the change in the wire's length and cross-section, they cause changes in electrical parameters (current and voltage) in an appropriate circuit.

The elastic captor for measuring force F , shown in the figure below, serves as the support for the R1, R2, R3, and R4 strain gauges, which are usually connected in a Wheatstone bridge circuit. As a consequence of the displacements of the points under the strain gauge's support, variations in the lengths (and cross-sections) of the conductive wires occur, and implicitly, variations in electrical resistance.



A.3. Purpose of the Application

This application presents the analysis of the displacement, strain, and stress fields within the elastic captor's structure. The analysis is performed to establish the zones with maximum deformations for applying the strain gauges and, on the other hand, to highlight that the material's yield limit is not reached under maximum load conditions. The presented captor, with the following dimensions: $R_e = 60$ mm, $R_i = 50$ mm, $R_l = 20$ mm,

$L = 120$ mm, $a = 90$ mm, $b = 60$ mm, $c_1 = 76$ mm, $c_2 = 45$ mm, $c_3 = 25$ mm, $g = 2$ mm, $\phi_1 = 10$ mm, $\phi_2 = 30$ mm, is designed to operate in the force range of 5-10 kN, applied to it through direct contact on the front surfaces of the counterbores with the adjacent loading parts. The material of the elastic element is OLC 45 steel, hardened and tempered to 250-270 HB.

B. PREPARING THE ANALYSIS MODEL

B.1. Defining the Analysis Model

To prepare the finite element analysis model associated with the application above, it is necessary to identify:

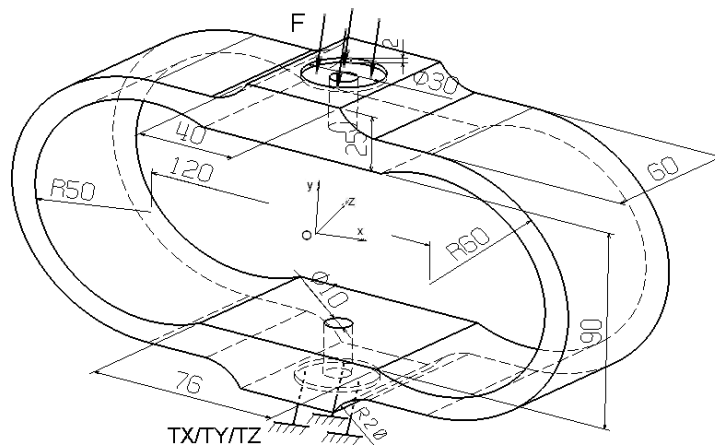
- The geometric shape and dimensions,
- Constraints induced by the connections with adjacent elements,
- External loads,
- Material characteristics.

B.2. Description of the Analysis Model

The geometric shape and dimensions of the analysis model are identical to those of the captor's 3D structure and are presented in the figure above.

For the analysis model to have the same behavior as the real model, it is necessary to associate boundary conditions. In this case, these are displacement restrictions along the three directions of the general XYZ coordinate system, applied to the points on the surface of the elastic element's seating counterbore.

The analysis model's structure is loaded on the surface of the other counterbore with the distributed force equivalent to the captor's maximum load, $F = 10$ kN.



B.3. Establishing Material Characteristics

For the finite element analysis, the strength characteristics of the material, OLC 45 steel, hardened and tempered to 250-270 HB, are:

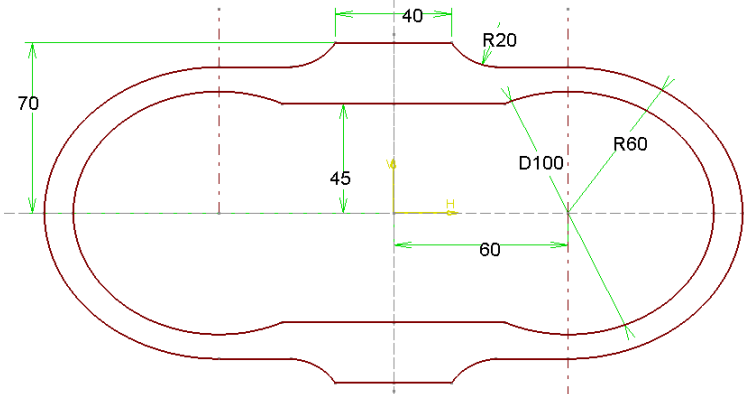
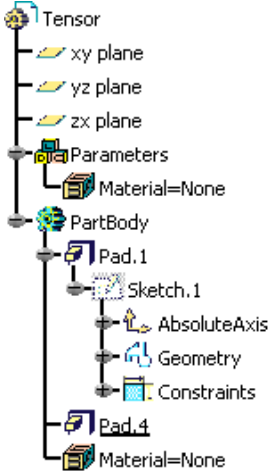
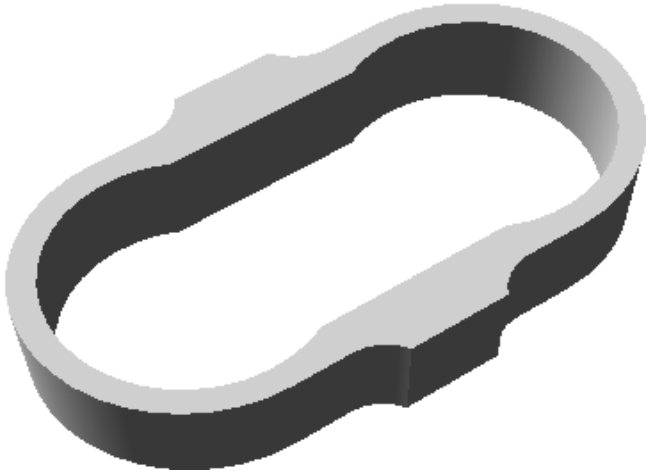
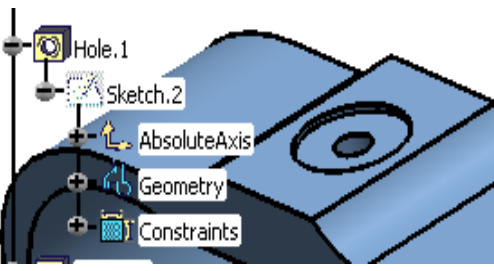
- Longitudinal modulus of elasticity, $E = 204000$ N/mm².
- Transverse contraction coefficient (Poisson's ratio), $\nu = 0,3$.

C. PREPROCESSING THE ANALYSIS MODEL

C.1. Geometric Modeling

Activate the solid generation module and set the unit of measurement for lengths

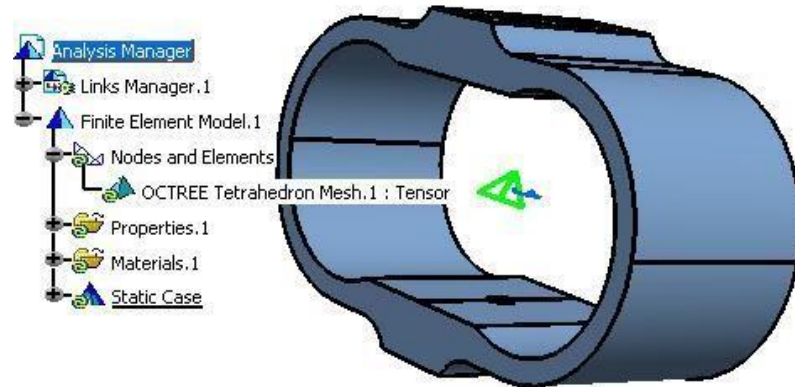
Start → Mechanical Design → Part Design → New part: New part name: Captor. Tools → Options ... → Options: Parameters and Measure; Units; Length, Millimeter (mm); ↵ OK.

Generate the reference sketch	
 (Sketcher) → xy plane →  (Profile) → [draw the closed contour of the frontal section] →  (Constraint) [successively introduce the dimensions by selecting the entities, using the Ctrl key for multiple selection] →  (Exit Workbench).	
Generate the solid model	
 (Pad) → Pad Definition: Length: 60, Selection: [select, Sketch.1], ↵OK.	
	
Generate the holes with counterbores	
 (Hole) → [select the face with the mouse] → Hole Definition: Extension ↓ Up to Next, Diameter: 10 mm, ☒ Normal to Surface, Positioning  (Sketch) → [position the hole center relative to the adjacent edges, using the  (Constraint) command],  (Exit Workbench) → Type: ↓ Counterbored, Diameter: 30 mm, Depth: 2 mm, ↵OK. [repeat this sequence for the other hole].	
C.2. Modeling the Material Characteristics	
[Select the PartBody specification in the tree structure] →  (Apply Material) → Library (ReadOnly): Metal, Steel → Apply Material, ↵OK. [Select Steel from the tree structure] → Properties: Feature Properties, Feature Name: OLC45; Analysis, Young Modulus 204000 N/mm², Poisson Ratio 0.3 → Apply, ↵OK.	

C.3. Finite Element Modeling

Activate the finite element analysis module

Start → Analysis & Simulation → Generative Structural Analysis → New Analysis Case: Static Analysis, ↵OK.



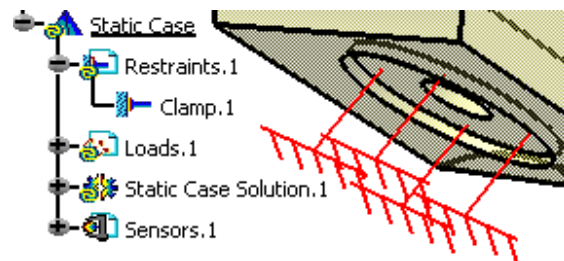
Observation

By activating the Generative Structural Analysis module, the solid model is implicitly set for discretization into linear tetrahedral finite elements (OCTREE Tetrahedron). Optionally, to modify the discretization characteristics (finite element type and its dimensions, globally or locally), the following sequence can be used. [Select with the mouse the specification OCTREE Tetrahedron Mesh: Part1, in the tree structure, or the symbol on the solid model structure 

C.4. Modeling Geometric Constraints

Introducing displacement restrictions

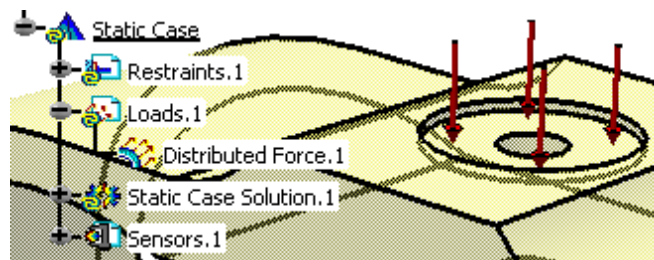
 (Clamp) → Clamp: Name Clamp.1, Supports [select the seating face with the mouse], ↵OK.



C.5. Modeling Loads

Introducing the force

 (Distributed Force) → Distributed Force: Supports [select the loaded face with the mouse]; Force vector: X 0 N, Y -10000 N, Z 0 N; ↵OK.



D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the Calculation Module

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

E. POSTPROCESSING THE RESULTS

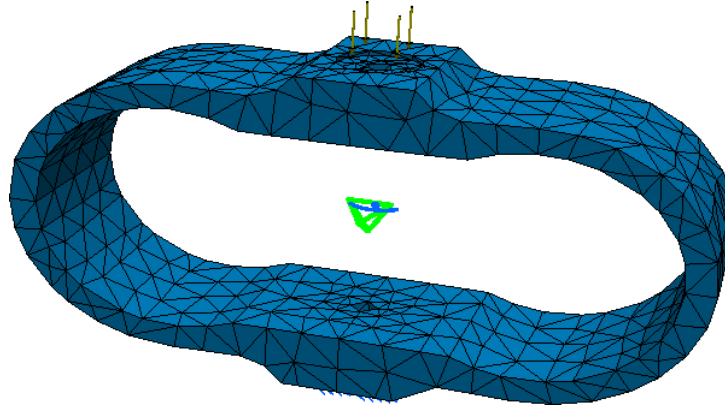
E.1. Visualizing the Deformed State/Animation



(Deformation).



(Animate) → Animate: Steps number: 5 (other parameters can be modified), ⌂Close.



Observation

To modify the displacement amplitude in the animation for improved suggestiveness, the following sequence can be followed:

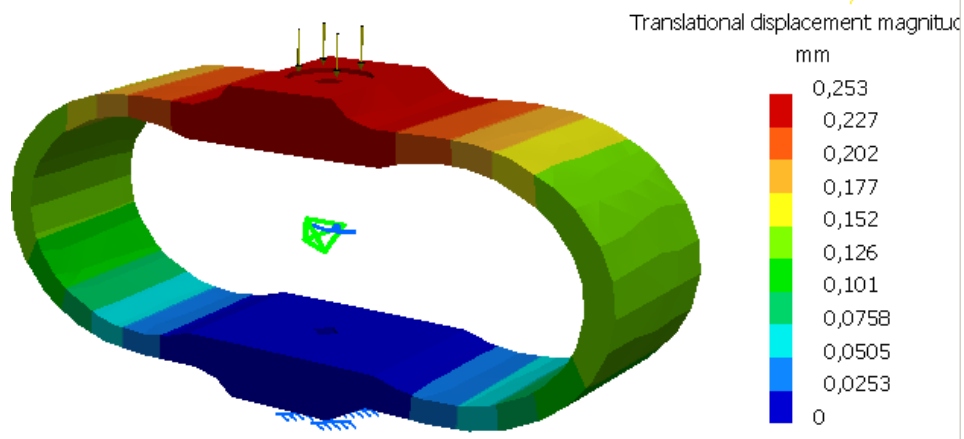


→ Amplification magnitude: Factor: 1 (other parameters can also be set), ⌂OK.

E.2. Visualizing the Displacement Field



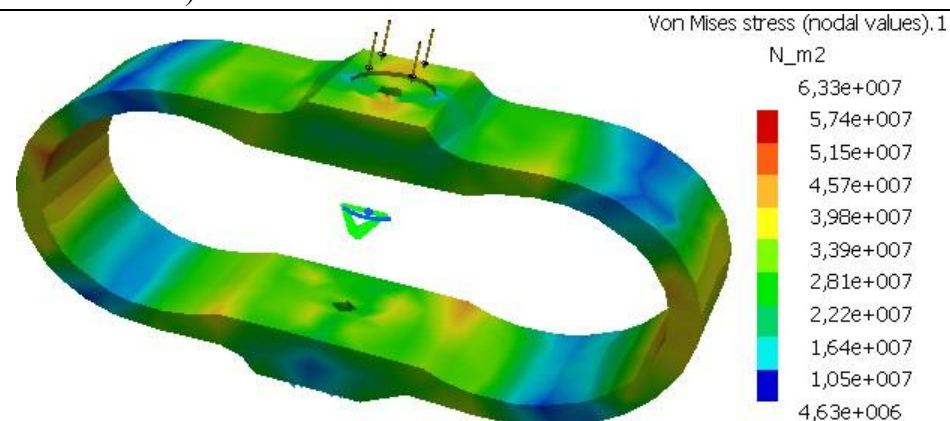
(Displacement).



E.3. Visualizing the Stress Field



(Von Mises Stress).



E.4. Generating the Analysis Report

 (Generate Report) → Report Generation: [select the analyzed case, Static case], ↵OK → [select the resulting .mht file for viewing (see Annex 1)].

F. CONCLUSIONS

The performance of an elastic captor for a tensiometric transducer is largely determined by its dimensions and configuration, which must meet requirements related to strain and stress states.

To obtain the highest possible sensitivity of the captor, it is necessary for the electro-resistive sensor (strain gauge) to be applied in an area where the deformations are quasi-constant, with values appropriate for the characteristics of the strain gauge used. In this case, this condition is evaluated by analyzing the displacement field in figure E.2. Thus, it is observed that in the lateral curved zones of the elastic element, the displacements in the direction of the measured force are quasi-constant with values of 0.07-0.09 mm (the central part of the area marked in green). On the other hand, analyzing the stress fields (figure E.3), they are localized in the same zones. In addition, it is observed that the maximum equivalent Von Mises stress (63.3 MPa) is below the admissible yield limit for the material used.

Computer-aided design of the elastic element is done iteratively. Starting from an initial structure, with the shape and dimensions established by classical methods, the finite element analysis model is prepared. After solving it and analyzing the results, if the limit conditions (of strength and/or displacement) are exceeded, modifications are made to the dimensions or even the shape, after which the new finite element model is re-analyzed. The process continues in order to approach the optimal structure (e.g., minimum mass) while respecting the strength and displacement conditions.

G. EXERCISE

Perform a finite element analysis of the elastic captor, shown below, which is a component of a tensometric transducer (load cell) for a scale. It is made of aluminum alloy (duralumin) and is loaded with a force of $F = 1\text{ kN}$.

