



FINITE ELEMENT METHOD

AEF-A.2.6

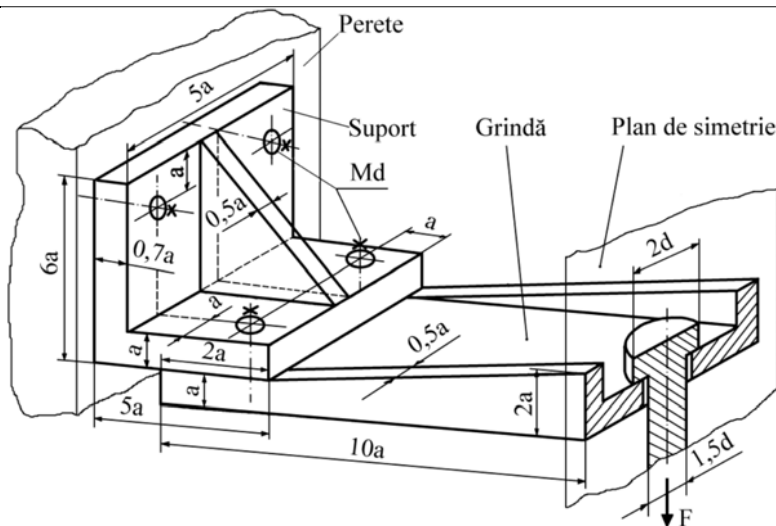
LINEAR STATIC ANALYSIS OF THE COMPONENT OF ASSEMBLED STRUCTURES

INTRODUCTION

Mechanical support assemblies are characterized by increased stiffness; their elastic deformation being reduced. The identification of stress and deformation fields for such structures can be done by finite element analysis, either for their component parts separately or for the whole structure, and the links between the component elements. The study of the tension and displacement fields in the mechanical assemblies of the components is mainly carried out in cases where some of them are obviously the most demanded, and on the other hand, the assembly areas are of no functional interest. The *Advanced Meshing Tools* and *Generative Structural Analysis* software modules of the CATIA environment allow the mechanical behavior of assembled mechanical structures to be simulated both by analyzing separate constituent elements and by assembling patterns with idealized elements implemented in these modules.

DESCRIPTION OF THE APPLICATION

The figure below shows the support structure consisting of two bearing pieces, called Support, and a beam bearing a weight F . The support part has the form of a central-ribbed L and four holes with a milling of settlement. The inner edges of the component parts are with radial connections r . The support elements are fixed to the side walls by a set of two M_d screws; the Beam element fixes why two supports are supported by two other M_d screws.



PURPOSE OF THE APPLICATION

In this application we present the study of the constructive elements of an assembled mechanical structure, considered as independent entities. To begin with, the analysis algorithm of the element named Support ($a = 10 \text{ mm}$, $d = 8 \text{ mm}$, $r = 0,2a$) is used to determine the displacement and tension fields, and the analysis of the Grind element is done as an exercise. These elements are made of steel, OL37. The bulk weight supported by the Beam element is $F = 24000 \text{ N}$.

FEA Application A.2.6

STATIC ANALYSIS OF THE COMPONENT ELEMENTS OF ASSEMBLED MECHANICAL SUPPORT STRUCTURES

A. DEFINING THE APPLICATION

A.1. Introduction

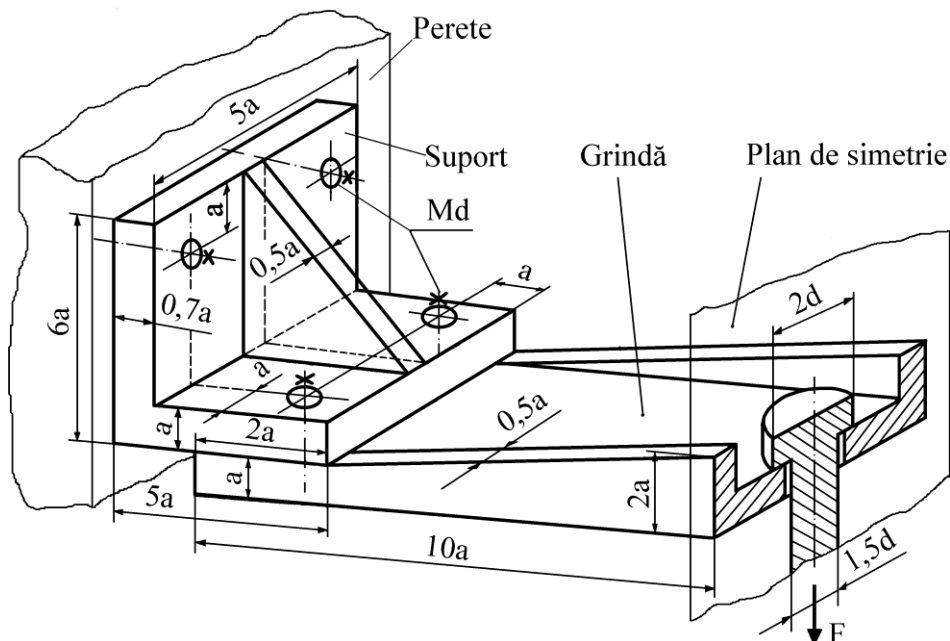
Assembled mechanical support structures, in most practical cases, are characterized by increased stiffness, their elastic deformations being small. The identification of stress and strain fields for such structures can be done by finite element analysis, either for their component parts separately or for the entire structure, also considering the connections between the component elements.

The study of stress and displacement fields in assembled mechanical structures, broken down by components, is primarily carried out in cases where some of them are clearly the most stressed and, on the other hand, the assembly areas are not of functional interest.

The Advanced Meshing Tools and Generative Structural Analysis software modules of the CATIA environment allow the simulation of the mechanical behavior of assembled mechanical structures, both by analyzing separate constituent elements and assemblies with connections modeled by idealized elements implemented in these modules.

A.2. Description of the Application

The figure below shows the support structure composed of two support pieces, called Support, and a beam from which a weight F is hung. The Support part has the form of an L with a central rib and four holes with counterbores (spotfaces). The inner edges of the component parts have radius fillets (r). The Support elements are fixed to the side walls by a set of two M d screws each; the Beam element is fixed to the two supports by two other sets of M d screws.



A.3. Purpose of the Application

This application presents the study of the constructive elements of an assembled mechanical structure, considered as independent entities. To begin, the analysis algorithm for the element named **Support** ($a = 10 \text{ mm}$, $d = 8 \text{ mm}$, $r = 0,2a$) is presented in order to determine the displacement and stress fields, with the analysis of the **Beam** element to be done as an exercise. These elements are made of steel, OL37. The weight supported by the assembly via the Beam element is $F = 24000 \text{ N}$.

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:

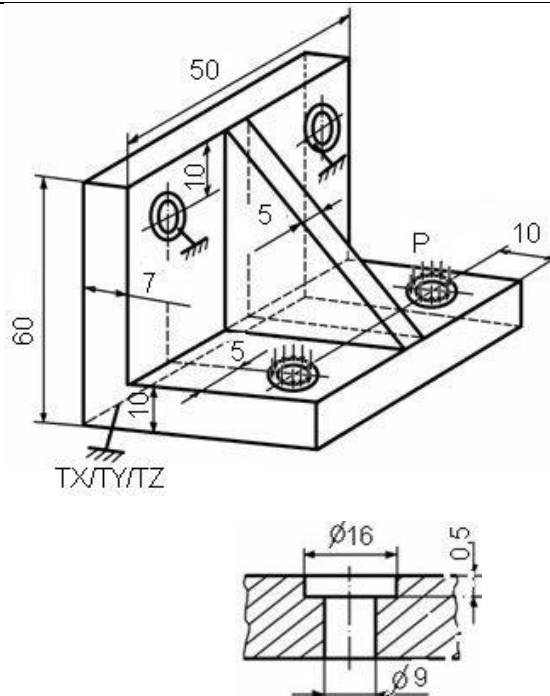
- The geometric shape and dimensions,
- Constraints induced by connections with adjacent elements,
- External loads,
- Material characteristics.

B.2 Description of the Analysis Model

The adjacent figure shows the detailed geometric shape and associated dimensions (the straight inner edges are filleted with $r = 0.2a$).

Simulating the wall fixation of the Support element is done by canceling the three possible translations of the points on the seating surfaces (counterbores) for the fixing screw heads and on the lower supporting edge.

The external loading is carried out by means of a force $P = F/4 = 6000 \text{ N}$, distributed on the seating surfaces (counterbores) of the Support-Beam fixing screws.



B.3. Establishing Material Characteristics

For the finite element analysis, the strength characteristics of the material, OL 37 steel, are:

- Longitudinal modulus of elasticity, $E = 204,000 \text{ N/mm}$.
- 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

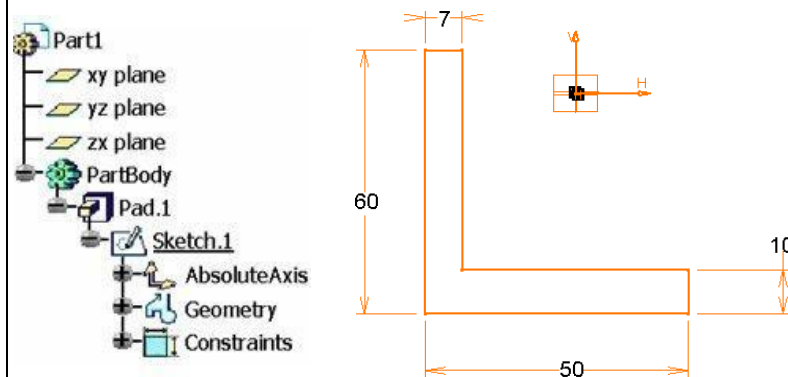
Start → Mechanical Design → Part Design → **New part**: New part name: Suport.

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

↵OK.

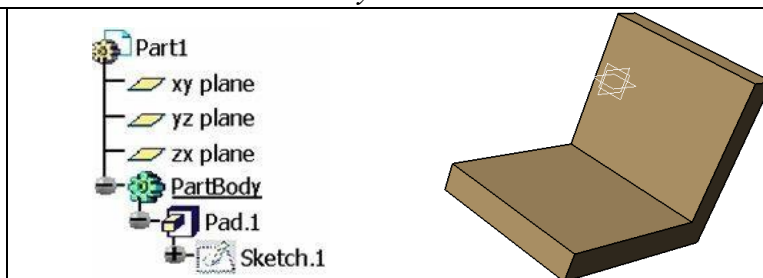
Generate the reference sketch

(Sketcher) → xy plane → (Profile) → [the closed contour of the frontal section with the mouse] → (Constraint) [successively introduce the dimensions by selecting the lines] → (Exit Workbench).



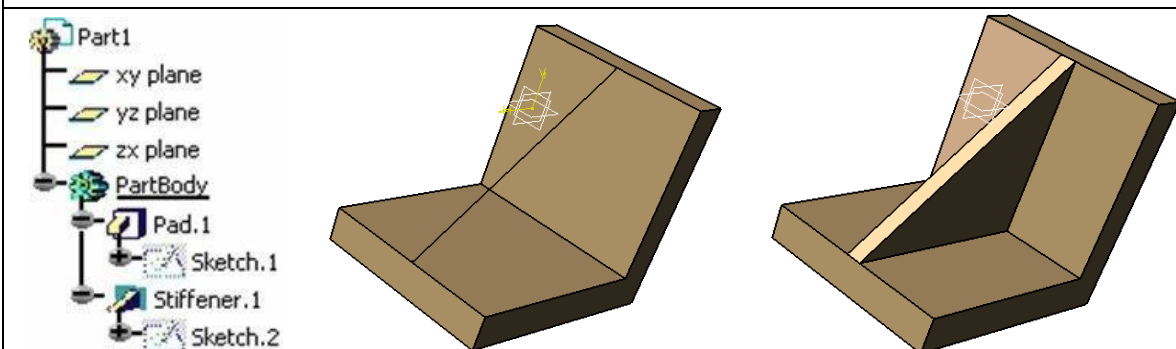
Generate the body

(Pad) → **Pad**
Definition: Length: 25,
Selection: [select Sketch.1
with the mouse],
☒ Mirrored extent, ↵OK.



Generate the rib

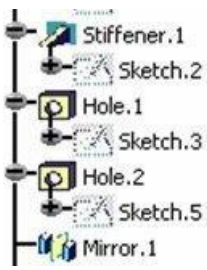
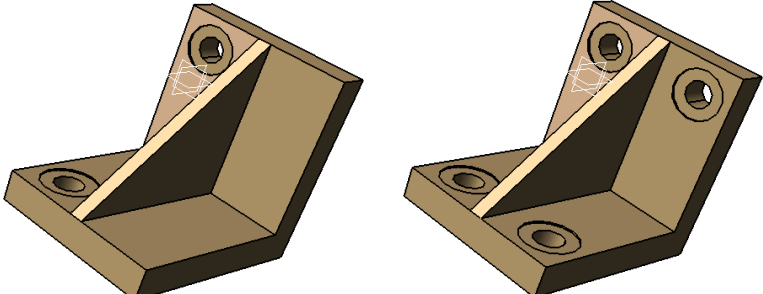
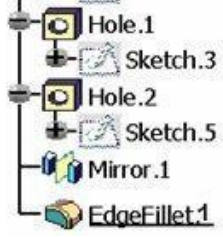
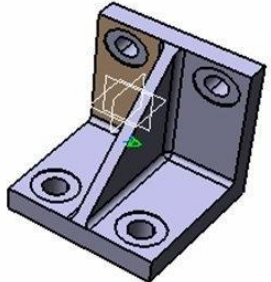
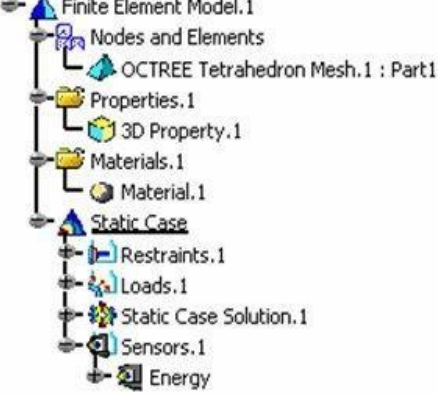
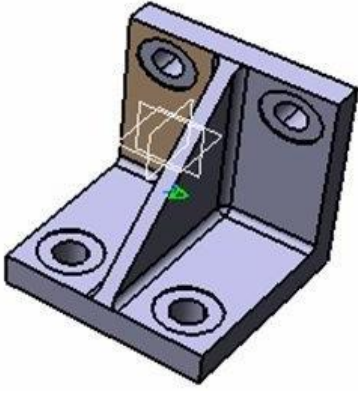
(Sketcher) → xy plane → (Line) → [draw the reference line of the rib outside the Sketch.1 area] → [select pairs of common points, using the Ctrl key] → (Constr. Def. in Dialog Box) → **Constraint Definition:** ☒ Coincidence, ↵OK → (Exit workbench).
 (Stiffener) → **Stiffener Definition:** Thickness: 4mm, ☒ Mirrored extent, Selection: [select Sketch.2 with the mouse], ↵OK.



Generate the holes with counterbores

(Hole) → [select the face on which the hole is generated with the mouse] → **Hole Definition:** Extension $\perp$ Up to Next, Diameter: 8 mm, ☒ Normal to Surface, Positioning Sketch → [position by dimensioning the hole center relative to the adjacent edges, using the (Constraint)], (Exit Workbench) → Type: $\perp$ Counterbored, Diameter: 15mm, Depth: 0,5mm, ↵OK. [Repeat this sequence for the other hole on the same side of the rib].

[Select holes, Hole.1 and Hole.2, with the mouse, using the Ctrl key] → (Mirror) → **Mirror Definition:** Mirroring element: [select the mirroring plane with the mouse, xy plane], ↵OK.

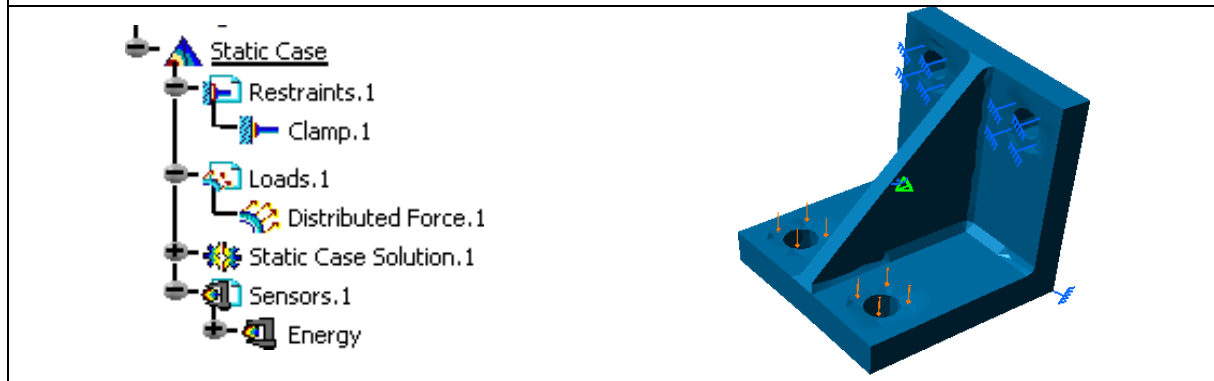
	
<i>Generate the fillets</i>	
[Select inner edges with the mouse, using the Ctrl key] →  (Edge Fillet) → Edge Fillet Definition: Radius: 2mm, Object(s) to fillet: 6 Edges, ↵OK.	 
C.2. Modeling the Material Characteristics	
[Select the PartBody specification] →  (Apply Material) → Library (ReadOnly): Metal, Steel, Apply Material; ↵OK. [Select the Steel specification from the tree structure] → Properties: Feature Properties, Feature Name: OL37; Analysis, Young Modulus 204000 N/mm², Poisson Ratio 0,3; Apply; ↵OK.	
C.3. Modeling the Finite Elements	
<i>Activate the finite element analysis module</i>	
Start → Analysis & Simulation → Generative Structural Analysis → New Analysis Case: Static Analysis, →OK.	 
Observation By activating the Generative Structural Analysis module, the solid model will be implicitly discretized into linear tetrahedral finite elements (OCTREE Tetrahedron). Optionally, to modify the discretization characteristics (the finite element type and its dimensions, globally or locally), the following sequence can be followed:[select with the mouse the specification OCTREF Tetrahedron Mesh: Part1, in the tree structure, or the symbol on the solid model structure ] → OCTREE Tetrahedron Mesh → [the necessary modifications will be made in this window].	
C.4. Modeling Geometric Constraints	
<i>Introducing displacement restrictions</i>	
 (Clamp) → Clamp: Supports [select with the mouse the counterbore faces for the screw head support and the supporting edge on the wall, using the Ctrl key], ↵OK.	



C.5. Modeling Loads

Introducing the forces

 (Distributed Force) → **Distributed Force**: Supports [select with the mouse the counterbore faces under load, using the Ctrl key]; Force vector X 0N, Y – 6000N, Z 0N; ↵OK.



D. SOLVING THE FINITE ELEMENT MODEL

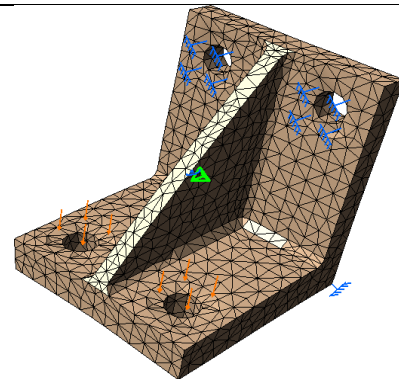
D.1. Launching the Calculation Module

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

E. POSTPROCESSING THE RESULTS

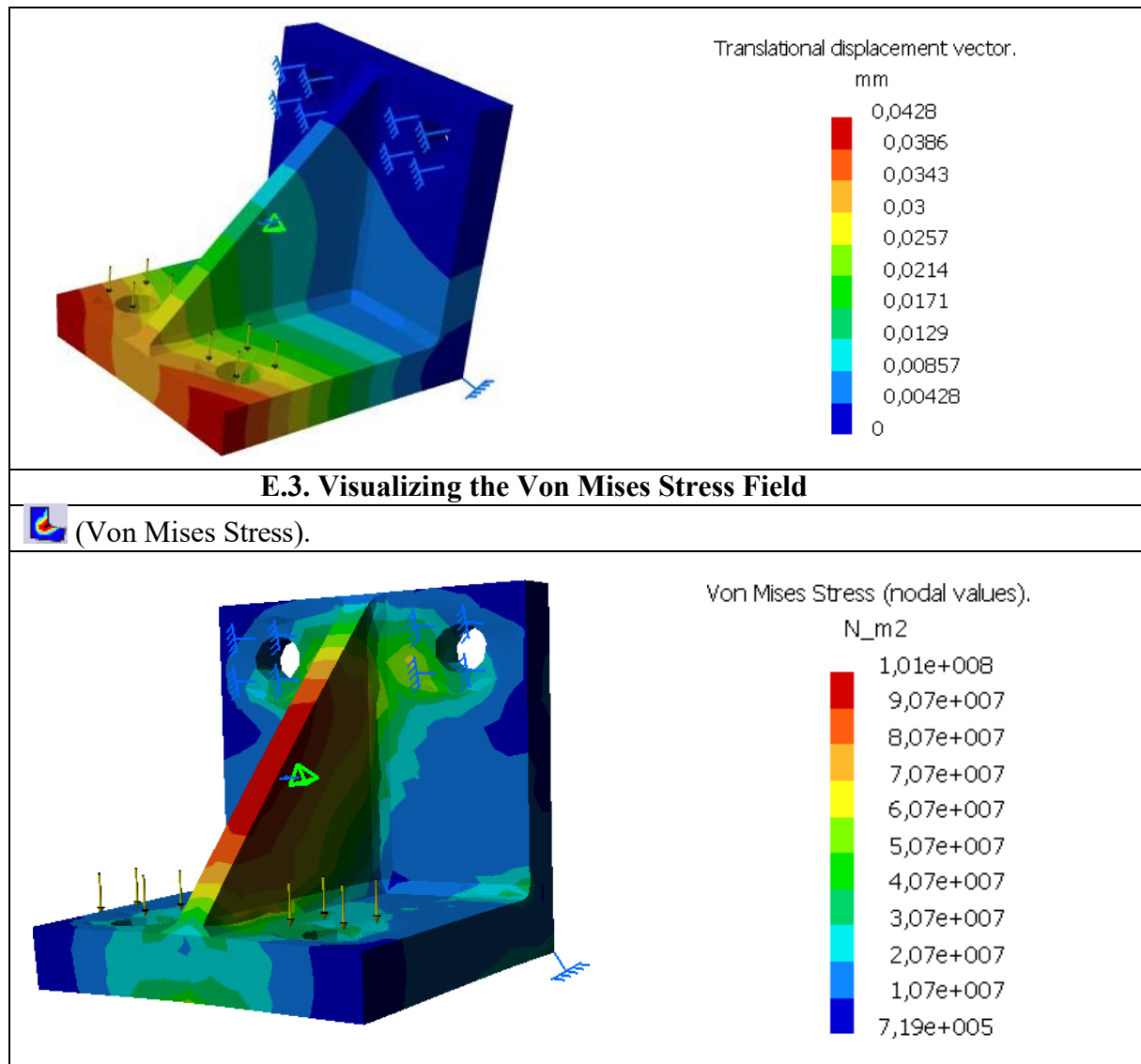
E.1. Visualizing the Deformed State/Animation

 (Deformation).
 (Animate) → **Animate**: Steps number: 20 (other parameters can be modified), ↵Close.
Observation
 To modify the displacement amplitude in the animation, follow the sequence
 → **Amplification magnitude** [adopt the desired scale factor], ↵OK.



E.2. Visualizing the Displacement Field

 (Displacement).



F. CONCLUSIONS

From the analysis of the displacement and stress fields, it is evident that the analyzed element is a rigid structure (maximum displacements are small, 0.0428 mm) with a maximum equivalent Von Mises stress of 96.82 MPa, in the rib area. It is observed that this value, for the OL37 material (yield strength approx. 330 MPa), is admissible, being much lower than the yield stress. If the maximum stress resulting from the analysis is higher or much lower than the admissible stress, the structure is under-dimensioned or, respectively, greatly over-dimensioned, and dimensional and/or shape modifications are required.

G. EXERCISE

Determine the displacement and stress fields for the Beam element, also made of OL37, and presented in the figure in subsection A.2.