



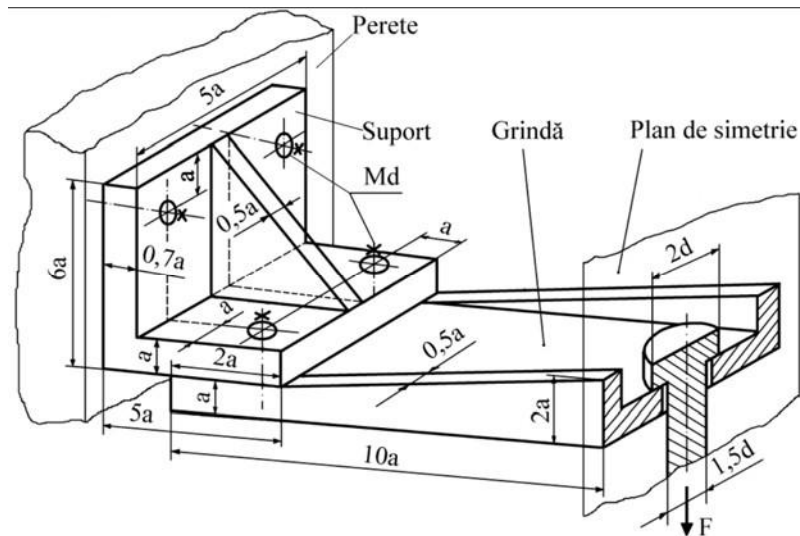
LINEAR STATIC ANALYSIS OF MECHANICAL ASSEMBLIES

INTRODUCTION

Mechanical structures assembled, in most practical cases, are characterized by increased stiffness, their elastic deformations being reduced. The identification of stress and deformation fields for such structures can be achieved by finite element analysis either for its component parts separately or for the entire structure, considering also the links between the component elements. The study of tension and displacement fields in assembled mechanical structures is carried out, in particular, in cases where overloaded components are not highlighted and the assembly areas are of 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, it is intended to study the above-mentioned mechanical structure consisting of several assembled elements in order to determine the values of the displacements and stresses in the structures of the component parts, taking into account the connections between them. The geometry of the *Support* and *Beam* elements is determined considering $a = 10 \text{ mm}$, $d = 8 \text{ mm}$, $r = 0,2 a$. These elements are made of steel, OL37. The solid model of the *Support* element was generated in application no.2.6 and can be downloaded as a .part extension file. It is also possible to take over the solid beam file of the Grind element, if the exercise in this application has been solved, or following the steps below. Following the geometric assembling of solid models, the finite element analysis model will be developed. The bulk weight supported by the Beam element is $F = 24000 \text{ N}$.

APPLICATION FEA-A.2.7

STATIC ANALYSIS OF ASSEMBLED SUPPORTING MECHANICAL STRUCTURES

A. APPLICATION DEFINITION

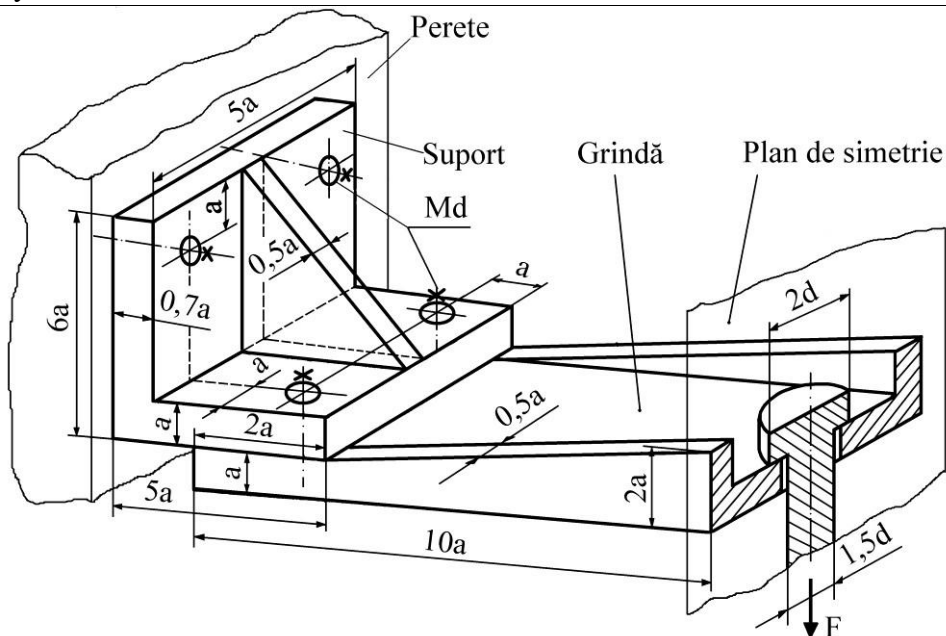
A.1. Introduction

Assembled mechanical structures, in most practical cases, are characterized by increased rigidity, their elastic deformations being reduced. The identification of the stress and deformation fields for such structures can be achieved by finite element analysis either for the component elements of the structure separately or for the entire structure, also considering the links between the component elements. The study of the stress and displacement fields in assembled mechanical structures is carried out especially in cases in which overloaded component elements are not highlighted and the assembly areas present interest from a functional point of view. The modules Advanced Meshing Tools and Generative Structural Analysis of the CATIA environment allow the simulation of the mechanical behavior of assembled mechanical structures both by analyzing certain constituent elements separately and as assemblies with connections modeled by idealized elements implemented in these modules.

A.2. Description of the application

The figure below shows the supporting structure composed of two bearing parts, called Support, and a beam from which a weight F is hung. The Support part has the shape of an L with a central rib and four holes with counterbore.

The inner edges of the component parts have fillets of radius 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. Scopul aplicației

In this application, the aim is to study the above mechanical structure consisting of several assembled elements in order to determine the values of the displacements and stresses in the structures of the component parts, taking into account the connections between them. The

geometry of the Support and Beam elements is determined considering: $a = 10 \text{ mm}$, $d = 8 \text{ mm}$, $r = 0.2a$. These elements are made of steel OL37. The solid model of the Support element was generated within Application no. 6 and can be imported as a file with the extension .part. It is also possible to import the solid model file of the Beam element, if the exercise in that application was solved, or following the steps below. After the geometric assembly of the solid models, the finite element analysis model will be created.

The weight supported by the assembly through the Beam element is $F = 24,000 \text{ N}$.

B. CREATION OF THE ANALYSIS MODEL

B.1. Definition of the analysis model

For the creation of the finite element analysis model associated with the above application it is necessary to identify:

- the geometric shape and dimensions,
- the restrictions induced by the connections with external elements,
- the internal links between the constituent elements,
- the external loadings,
- the material characteristics.

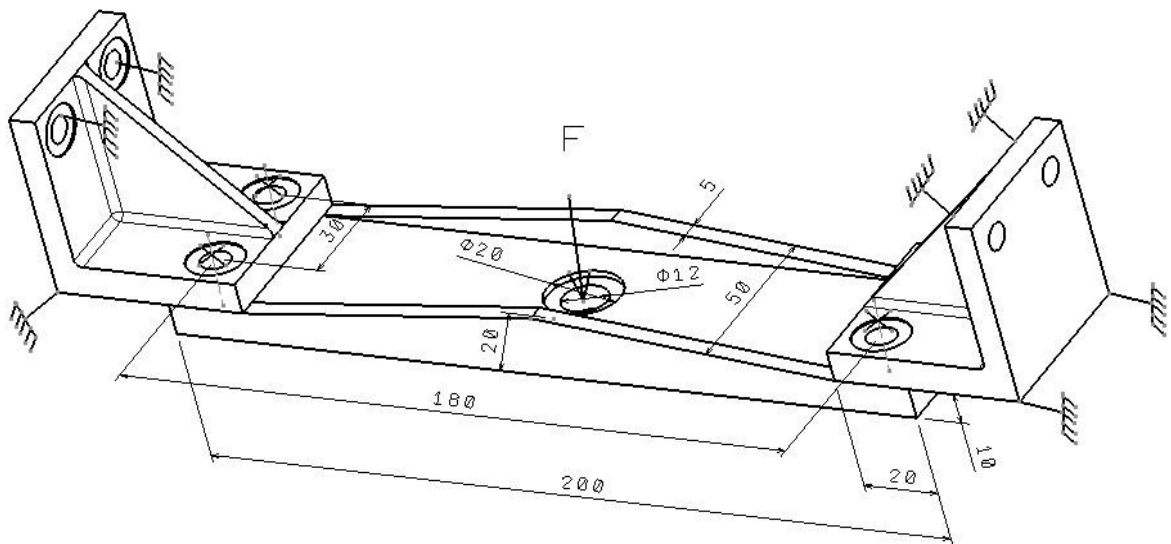
B.2 Description of the analysis model

The figure below shows the detailed geometric shape of the assembly elements (two Support elements and one Beam element) and the dimensions associated with the Beam element.

Simulation of the wall fixation of the Support elements is carried out, as in Application no. 6, by cancelling, for each of them, the three possible translations of the points on the seating surfaces of the fixing screw heads and on the lower support edge.

The connections physically created by threaded assemblies between the Beam and the two Support elements are modeled using idealized elements of indirect connection with rigid pretensioned bolt (Bolt Tightening Connection, see subchapter 1.2.2.2).

The external loading of the assembly is applied by a force distributed on the counterbored surface, in the center of the Beam element, for the placement of the central bolt, with the value $24,000 \text{ N}$.



B.3. Establishing the material characteristics

For the finite element analysis, the mechanical characteristics of material OL37 steel are:

- longitudinal elasticity modulus $E = 204,000 \text{ N/mm}^2$
- transverse contraction coefficient (Poisson) $\nu = 0.3$

C. PRE-PROCESSING OF THE ANALYSIS MODEL

C.1. Geometric modeling of the Beam element

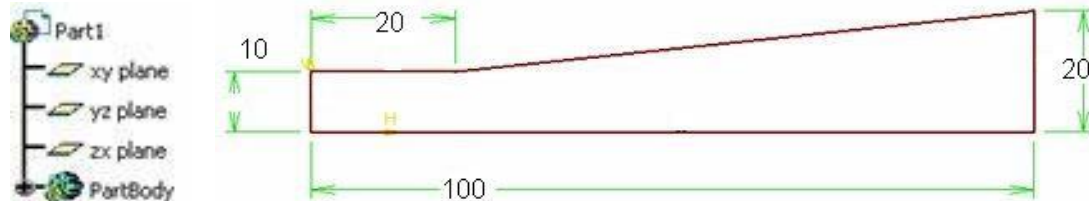
Activating the solid generation module and setting the unit of measurement for lengths

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

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

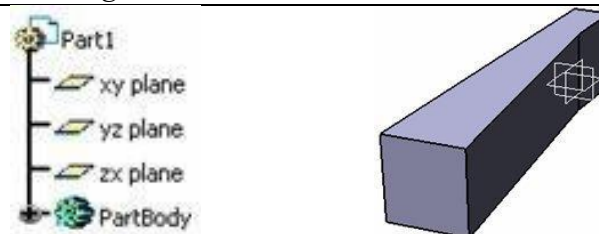
Generating the reference sketch

 (Sketcher) → xy plane →  (Profile) → [draw the closed semi-contour of the front view with the mouse] →  (Constraint) [enter dimensions successively by selecting lines with the mouse] →  (Exit Workbench).



Generating the solid model

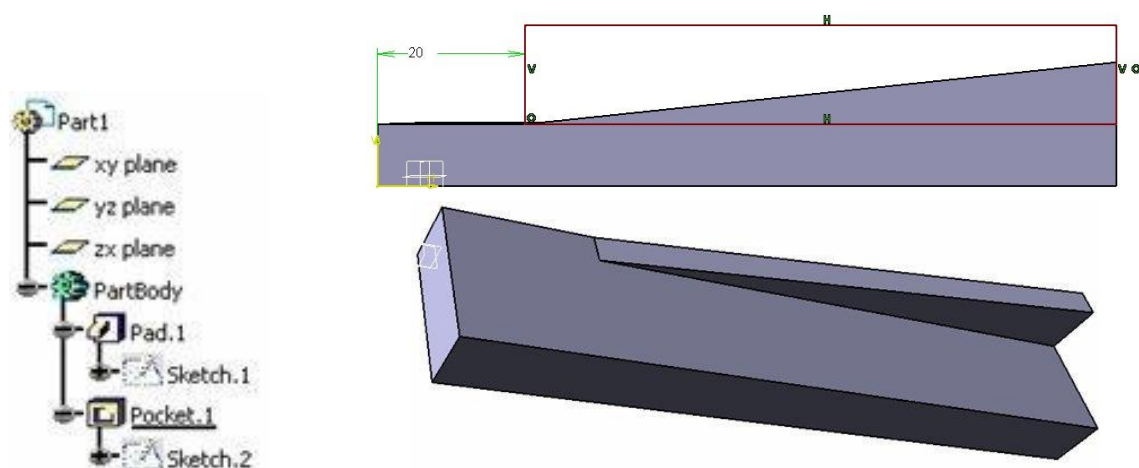
 (Pad) → **Pad Definition:** Length: 25, Selection: [select **Sketch.1** with the mouse], ☐ OK.



Generating the cutout

 (Sketcher) → xy plane →  (Rectangle) [draw the rectangle polyline with the mouse] →  (Exit workbench).

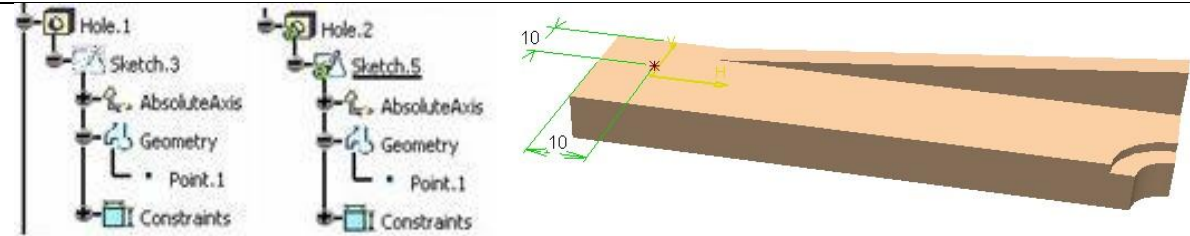
 (Pocket) → **Pocket Definition:** Selection: [select the rectangle polyline **Sketch.2** with the mouse], Depth: 20, Reverse Direction, ☐ OK.



Generating the holes

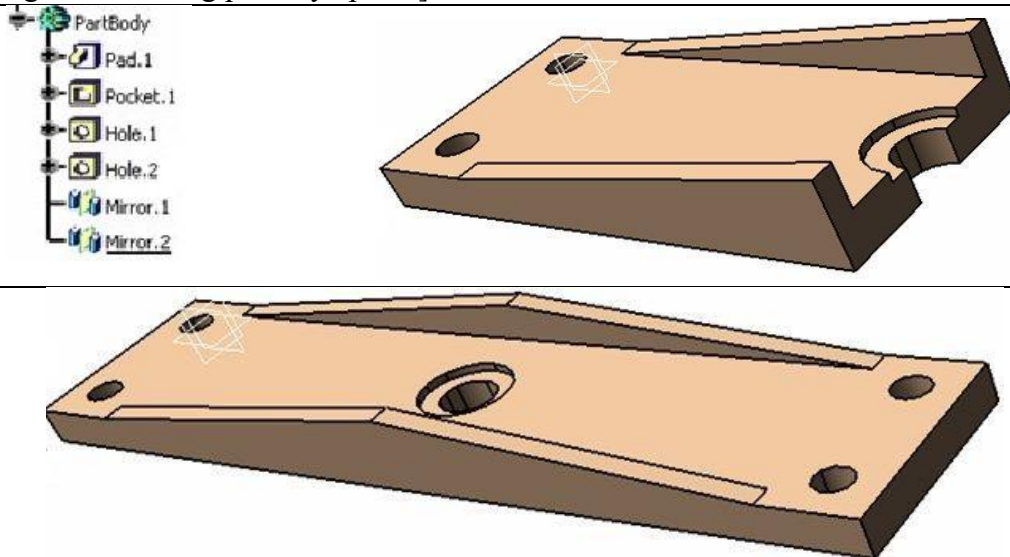
 (Hole) → [select the face where the hole is generated with the mouse] → **Hole Definition:** Extension Up to Next, Diameter: 8 mm, ☒ Normal to Surface, Positioning Sketch  → [position by dimensioning the hole center relative to adjacent edges (see fig. below), using the (**Constraint**) command ], (Exit Workbench)  → ☐ OK; [repeat this sequence for the other hole on the same side of the rib]

 (Hole) → [select the face where the hole is generated with the mouse] → **Hole Definition:** Extension Up to Next, Diameter: 8 mm,  Normal to Surface, Positioning Sketch  → [position the hole center in the corner opposite the origin (see fig. below), using the (Constraint) command ],  (Exit Workbench) → Type:  Counterbored, Diameter: 15mm, Depth: 0,5mm, OK.



Generating the beam body

[Select PartBody with the mouse] →  (Mirror) [select the mirroring plane, **xy plane**] → **Mirror Definition:** Mirroring element: xy plane, OK. [Repeat this sequence of commands, selecting the mirroring plane, yz plane].



Modeling 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.



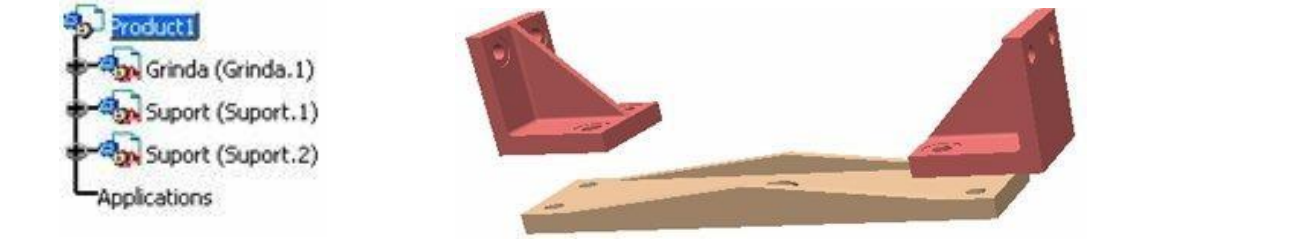
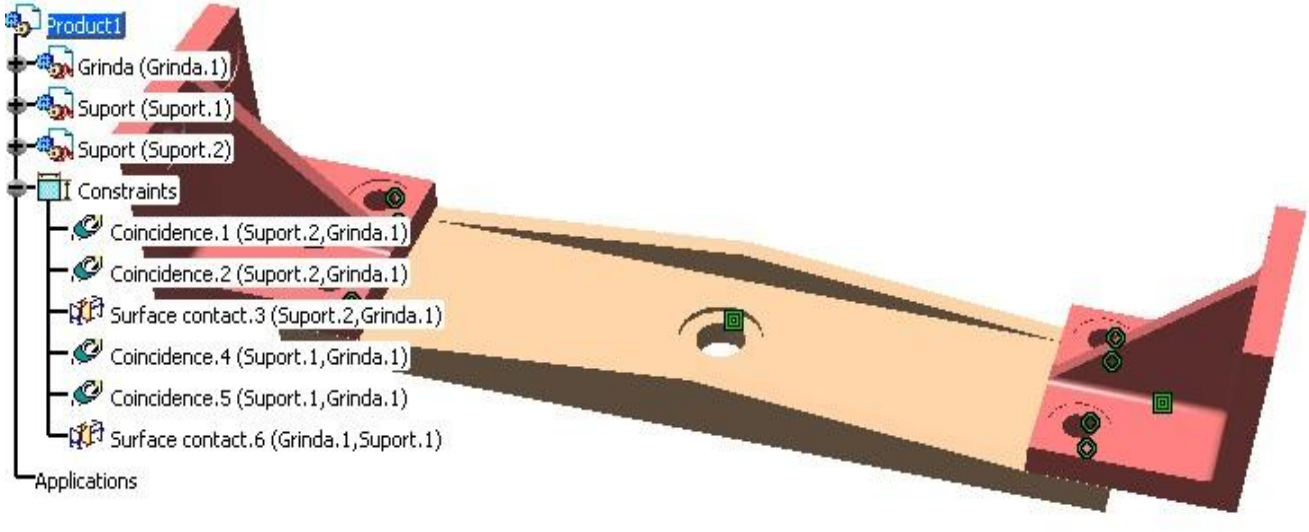
C.2. Geometric modeling of the assembly

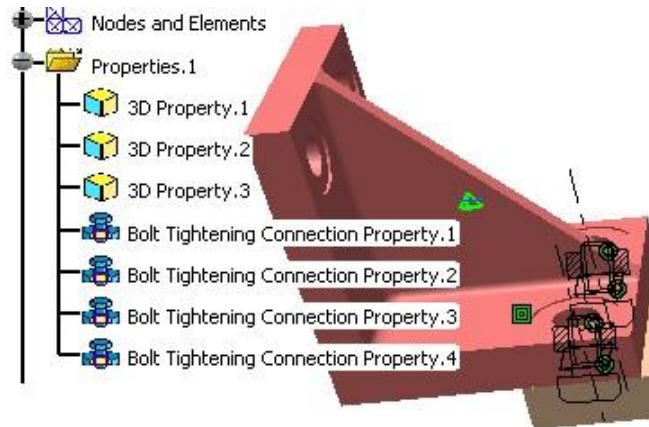
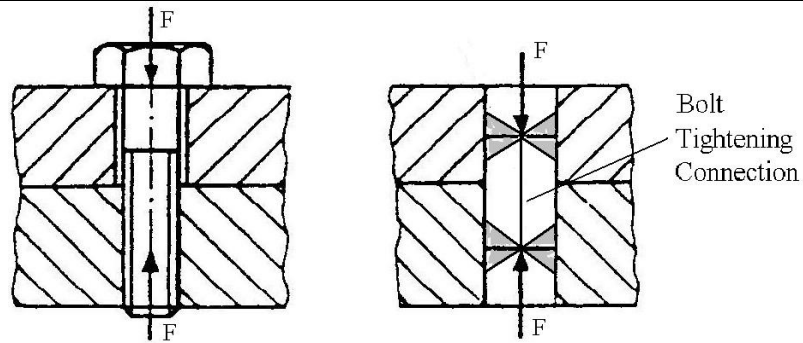
Activating the assembly generation module and loading component parts

Start → Mechanical Design → Assembly Design.

Insert → Existing Component → [activate Product1 specification] → [select path] → Grinda.CATPart → Open. [Repeat this sequence to load the file Suport.CATPart twice].

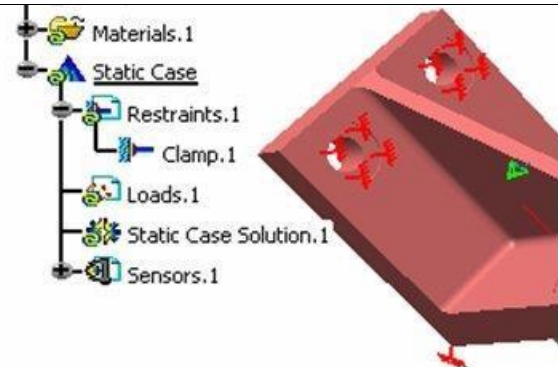
 (Manipulation) → **Manipulation Parameters:** [select the direction of manipulation in this window followed by the actual manipulation of a body to eliminate overlap], OK.


<i>Introducing assembly geometric constraints</i>
[Select the coincident plane surfaces on the Support and Beam with the mouse, using the Ctrl key] →  (<i>Contact Constraint</i>) →  (Update All). [Repeat this sequence for the other set of common Support-Beam plane surfaces]. [Select the coincidence cylindrical surfaces on the Support and Beam with the mouse, using the Ctrl key] →  <i>Coincidence Constraint</i> →  (Update All). [Repeat this sequence for the other three sets of common Support-Beam cylindrical surfaces].

C.3. Finite element modeling
<i>Activating the finite element analysis module</i>
Start → Analysis & Simulation → Generative Structural Analysis → New Analysis Case: Static Analysis, OK.
C.4. Modeling constraints
<i>Modeling bolt connections with pre-tension</i>
 (Bolt Tightening Connection Property) → Bolt Tightening Connection: Supports: [select the geometric restriction Coincidence.1(Suport.2, Grinda.q1), see fig. above], Tightening force: -100 N, Orientation: ↓ Opposite, OK. [Repeat this sequence for the other three Coincidence constraints].
Observation - To visualize the geometric constraints, activate the specification in the specification tree: Analysis Manager → Links Manager.1 → Link.1 → Ans Grinda support → Constraints. - The bolted assemblies in this subassembly are modeled with the surface-to-surface connection element Bolt Tightening Connection (see subchap. 1.2.2.2) which assumes the connection between two cylindrical surfaces acting on each other with the tightening force F (Tightening force). The orientation of this element is defined in the Orientation box and can be the same as the geometric constraint (Same) or opposite to it (Opposite).



Introducing displacement restrictions

 (Clamp) → **Clamp**:
Supports [select the face of the loaded counterbore with the mouse, using the **Ctrl** key], OK.



C.5. Modeling loads

Introducing the force

 (Distributed Force) → **Distributed Force**:
Supports [select the face of the loaded counterbore with the mouse, using the **Ctrl** key]; Force vector X 0N, Y 4000N, Z 0N; ☐ OK.



D. SOLVING THE FINITE ELEMENT MODEL

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

E.1. Visualizing the deformed state/Animation



(Deformation).



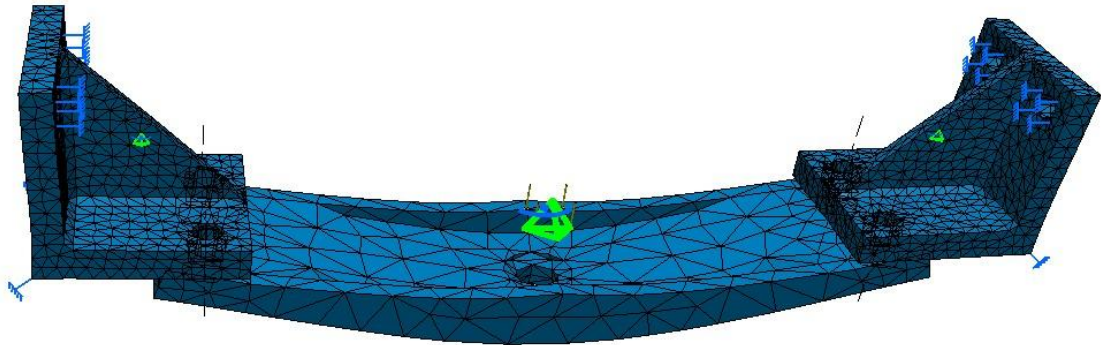
(Animate) → Animate: Steps number: 20 (other parameters can also be modified), Close.

observation

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



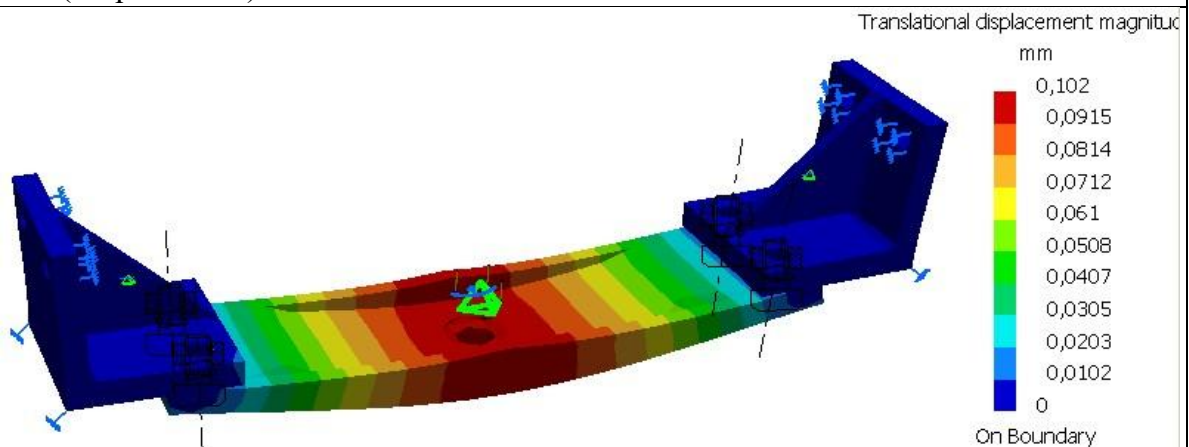
→ Amplification **magnitude**: Factor: 1 (other parameters can be set), OK.



E.2. Visualizing the displacement field



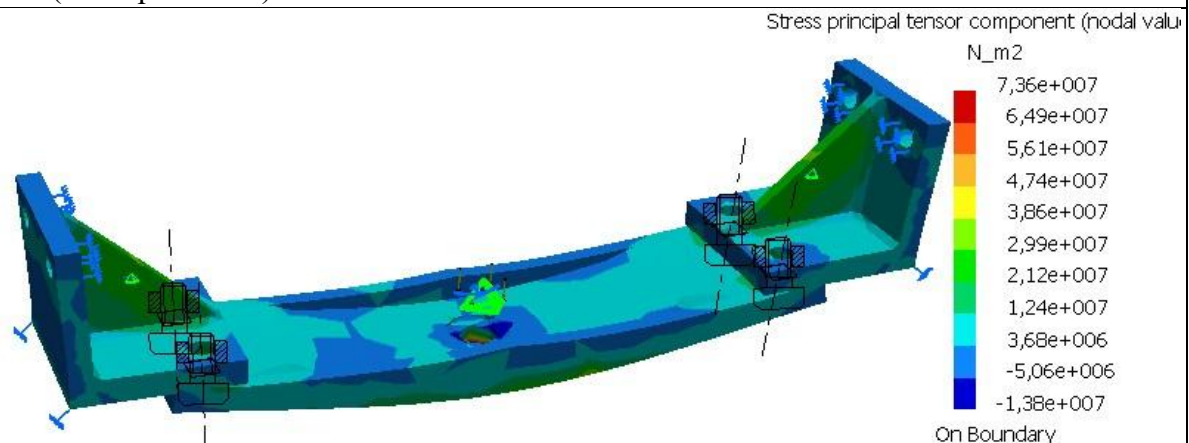
(Displacement).



E.3. Visualizing the stress field

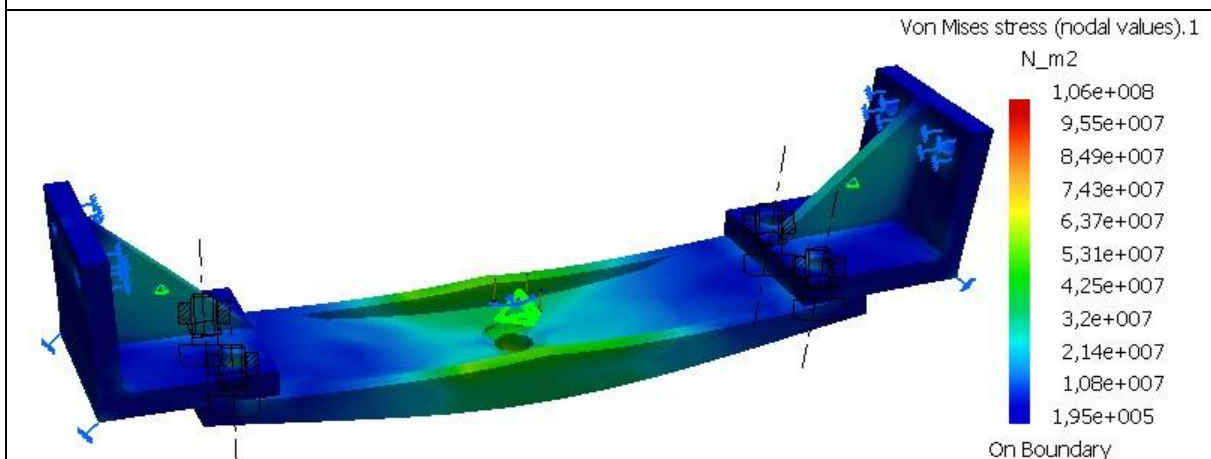


(Principal Stress).





(Von Mises Stress).



F. CONCLUSIONS

The analysis of the displacement and stress fields highlights that the analyzed subassembly is a quasi-rigid structure (reduced maximum displacements, 0.102 mm) with maximum principal stresses (73.6 MPa) and Von Mises stresses (106 MPa) in the area of the support rib. It is observed that these values, for the OL37 material with a yield strength of approx. 330 MPa, are acceptable, being much lower than the limit.

G. EXERCISE

Analyze the displacement and stress fields for the assembly in fig. A.2 considering the connections between the Beam and Support as ideal surface-to-surface assemblies, using the Fastened Connection element (see subchap. 1.2.2.2) and, additionally, considering the self-weight forces.