



FINITE ELEMENT METHOD

AEF-A.2.8

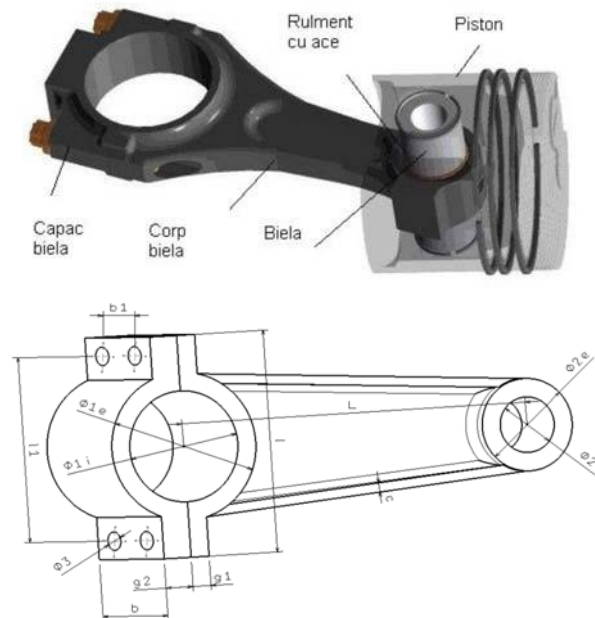
LINEAR STATIC ANALYSIS OF AN INTERNAL COMBUSTION ENGINE ROD

INTRODUCTION

Engine rods are important parts of internal combustion engines characterized by increased stiffness; their elastic deformation being reduced. Constructively, the rod subassemblies are composed of two pieces (body and lid) solidarized by means of assemblies with play-assisted screws; the clamping force of these screws has a decisive role in achieving the stiffness of the subassembly. Determination of stress and strain fields for such structures and the study of their variation with the change in the force of the screws are useful for determining the stiffness of the subassembly and the strength of its components.

DESCRIPTION OF THE APPLICATION

The figure below shows a piston-rod assembly from an internal combustion engine. The rod is made up of the body, cover and fastening screws. In the bore holes a single-block or a needle bearing forms the coupling with the bolt and the half-bearings that form the coupling with the crankshaft. Loads, as a consequence of the forces transmitted, are carried out in these camps with forces distributed according to the variation laws dependent on the type of the bearing and the adopted version.



PURPOSE OF THE APPLICATION

In this application we present the finite element analysis algorithm of a rod stressed by force loading $F = 3000 \text{ N}$ in the needle radial bearing areas. The rod cap is assembled to the stem body by a set of four M8 screws. The dimensions of the elements of the analyzed subassembly are as follows: $\phi_{1e} = 64 \text{ mm}$, $\phi_{1i} = 48 \text{ mm}$, $\phi_{2e} = 40 \text{ mm}$, $\phi_{2i} = 24 \text{ mm}$, $\phi_3 = 8 \text{ mm}$, $L = 160 \text{ mm}$, $l = 96 \text{ mm}$, $l_1 = 80 \text{ mm}$, $b = 40 \text{ mm}$, $b_1 = 20 \text{ mm}$, $c = 4 \text{ mm}$, $g_1 = 8 \text{ mm}$, $g_2 = 12 \text{ mm}$. The inner edges of the body and bead cap are with radial connections $r = 2 \text{ mm}$. The elements of the analyzed subassembly are made of OLC 45 steel, improved to 250-270 HB.

Application FEA-A.2.8

LINEAR STATIC ANALYSIS OF A CONNECTING ROD SUBASSEMBLY

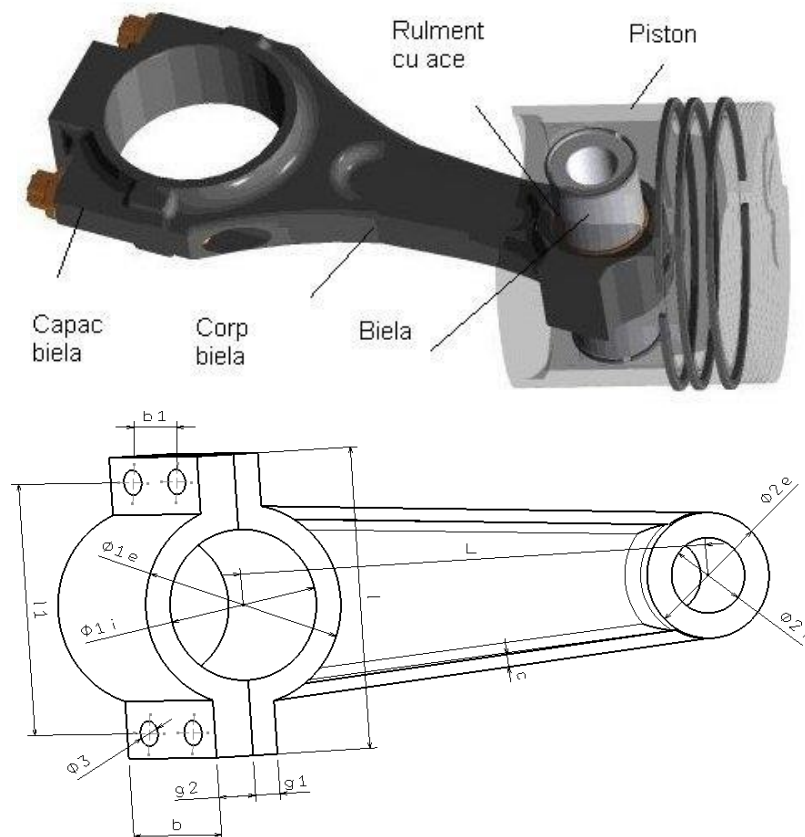
A. DEFINITION OF THE APPLICATION

A.1. Introduction

Connecting rod subassemblies are important parts of internal combustion engines characterized by increased stiffness, with reduced elastic deformations. Constructively, rod subassemblies are composed of two pieces (body and cap) joined by screws mounted with clearance. The tightening force of these screws plays a decisive role in achieving the stiffness of the subassembly. Determining the stress and strain fields for such structures and studying their variation with changes in the screw pre-tensioning force are useful for determining the stiffness of the subassembly and the strength of its components.

A.2. Description of the Application

The figure in the document presents a piston-rod assembly from an internal combustion engine. The rod subassembly is composed of the body, the cap, and fastening screws. A single-block bushing or a needle bearing is mounted in the rod bores, forming the coupling with the piston pin, and half-bearings (bearing shells) form the coupling with the crankshaft. Loads, as a consequence of the transmitted forces, are applied in these bearings as distributed forces according to variation laws dependent on the bearing type and the adopted design version.



A.3. Purpose of the application

This application presents the finite element analysis algorithm for a rod subassembly subjected to tension by a load of $F=3000$ N in the areas of the radial needle bearing and the rod cap bearing shell. The rod cap is assembled to the rod body by a set of four M8 screws. The dimensions of the elements of the analyzed subassembly are:

C. PRE-PROCESSING OF THE ANALYSIS MODEL

C.1. Geometric modeling of the rod body

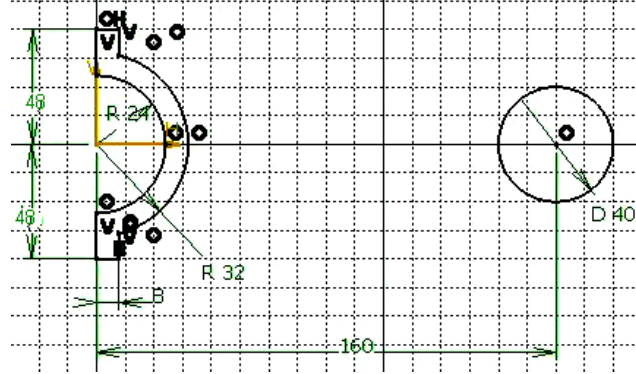
Activate solid generation and set units

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

Generating the reference sketch for the rod head and base:

(Sketcher) → xy plane → (Profile) → [draw the closed contours associated with the rod head and base] → (Constraint) [enter dimensions successively] → (Exit Workbench).

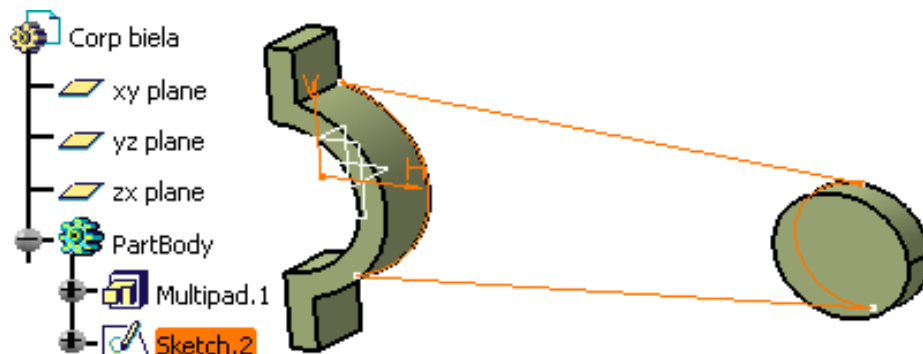
(Multi-Pad) → **Multi-Pad**
Definition: Domains [select Extension domains.1], Length 20, [select Extension domains.2], Length 10, OK.



Generating the rod body connection

(Sketcher) → xy plane → (Project 3D Elements) [select lines common to the two sketches] → (Line) [draw outer lines] → (Constraint) [make points coincident with the base, using Ctrl] → (Constraint) → **Constraint Definition**: *Coincidence*, ↵OK → [make points coincident with the base, using Ctrl] → (Constraint) → **Constraint Definition**: *Tangency*, ↵OK. [between outer line and rod head circle].

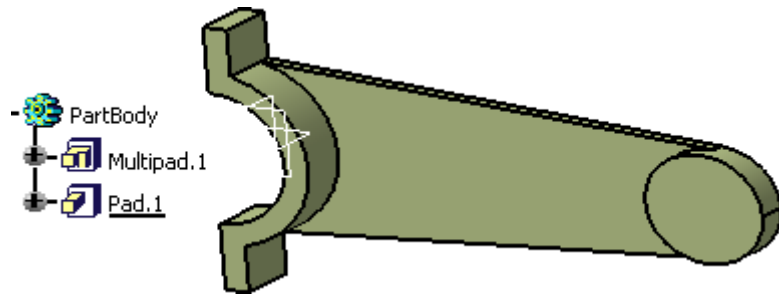
(Quick Trim) [delete the outer arc of the rod head circle] → (Exit Workbench).



Generating the rod body connection



(Pad) → **Pad**
Definition: Selection:
 [select Sketch.2],
 Length: 4, ↵ OK.



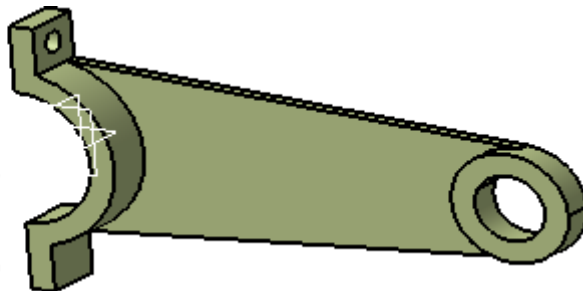
Generating holes



(Hole) → [select face] → **Hole Definition:** Extension ↓ Up to Next, Diameter: 24 mm, Normal to Surface, Positioning Sketch → [position center using Constraint], (Exit Workbench), ↵ OK.

[Repeat for the 8 mm hole in the rod head].

[Select the 8 mm hole] → (Mirror) → **Mirror Definition:** Mirroring element: [select yz plane], ↵ OK.



Generating stiffeners

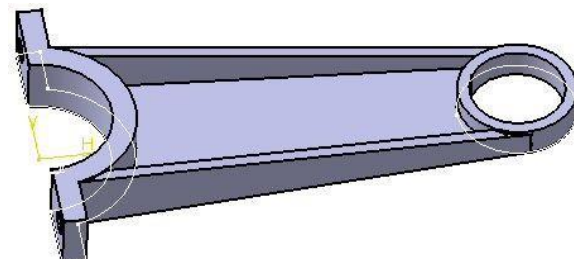
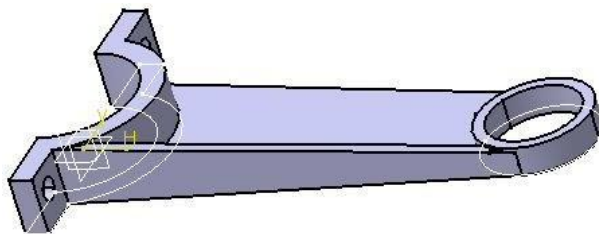
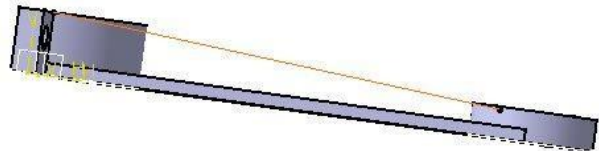


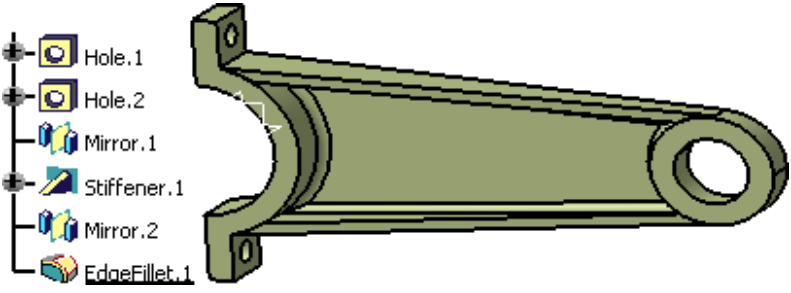
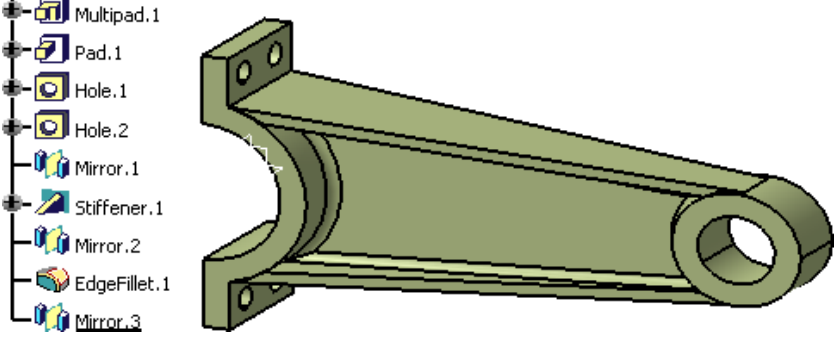
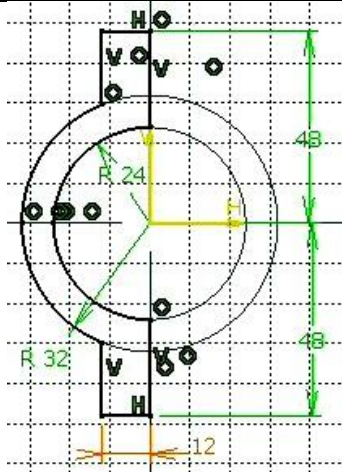
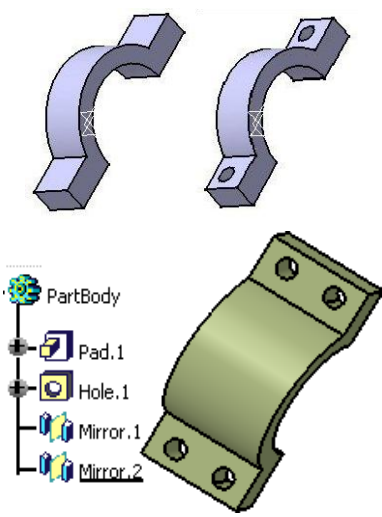
(Sketcher) → [select stiffener plane] → (Line) → [draw reference line outside the zone], (Constraint) [use Coincidence to join this line to the rod body structure] → (Exit Workbench).



(Stiffener) → **Stiffener Definition**
 Thickness: 4mm, Selection [select Sketch.2], ↵ OK.

[Select stiffener] → (Mirror) → **Mirror Definition:** Mirroring element: [select yz plane], ↵ OK.



Generating fillets	
[Select inner edges using Ctrl] → (Edge Fillet) → Edge Fillet Definition Radius: 3mm, Object(s) to fillet: 4 Edges ↵OK.	
Generating the full rod body	
[Select the rod semi-body PartBody] → (Mirror) → Mirror Definition: Mirroring element: [select zx plane], ↵OK.	
C.2. Obtaining the solid model of the rod cap	
Activating the solid generation module and generating the reference sketch of the lid	
Start → Mechanical Design → Part Design → New part: New part name: Capac. (Sketcher) → xy plane → (Profile) → [draw closed contour] → (Constraint) → (Exit Workbench).	
Generating the Cap Body	
(Pad) → Pad Definition Length: 20, Selection: [select Sketch.2], ↵OK. (Hole) → [select the face where the hole is generated] → Hole Definition: Extension ⊥ Up to Next, Diameter: 8 mm, Normal to Surface, Positioning Sketch → [position by dimensioning the hole center using the (Constraint) command], (Exit Workbench) → ↵OK. [Select the 8 mm diameter hole with the mouse] → (Mirror) → Mirror Definition: Mirroring element: [select mirroring plane, yz plane], ↵OK. [Select the semi-cap, PartBody, with the mouse] → (Mirror) → Mirror Definition: Mirroring element: [select mirroring plane, xy plane], ↵OK.	

C.3. Modeling Material Characteristics

[Select PartBody 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.



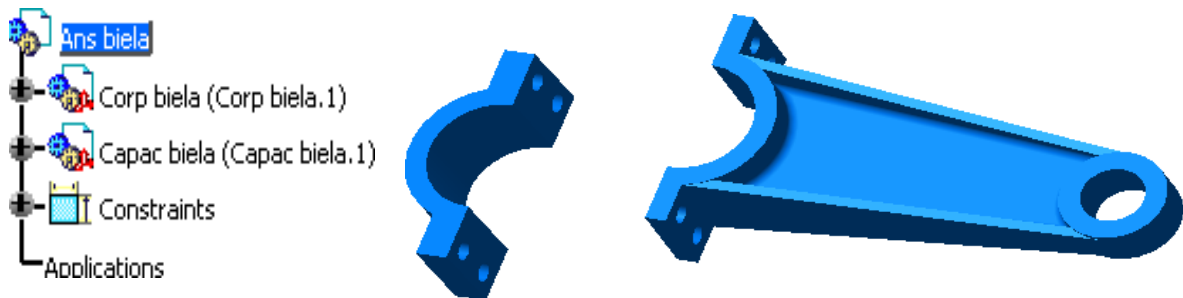
C.4. Geometric Modeling of the Assembly

Activating the assembly generation module and loading component parts

Start → Mechanical Design → Assembly Design.

Insert → Existing Component → [activate Product1 specification] → [select path] → Biela.CATPart → Open. [Repeat this sequence to load the Capac.CATPart file as well].

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

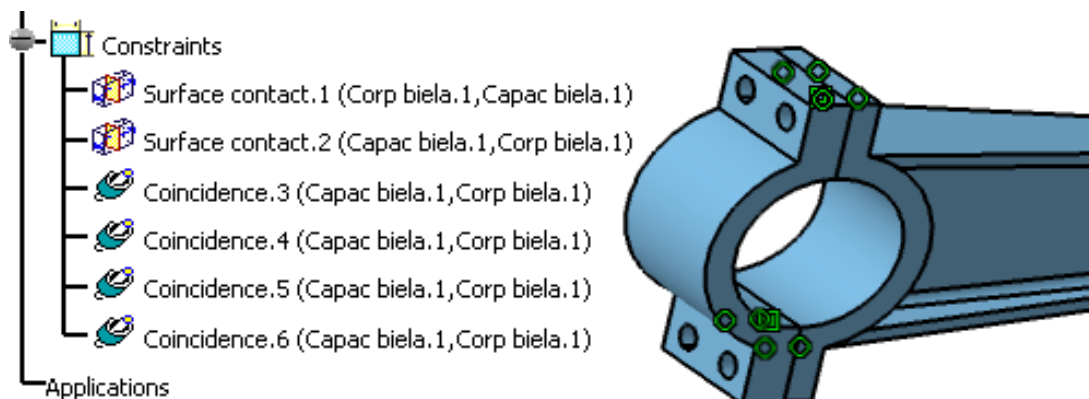


Introducing assembly geometric constraints

[Select the coincident plane surfaces of the rod and cap with the mouse, using the Ctrl key] → (Contact Constraint) → (Update All).

[Select a pair of coincident cylindrical surfaces of the rod and cap with the mouse, using the Ctrl key] → Coincidence Constraint → (Update All).

[Repeat this sequence for the other three pairs of cylindrical surfaces of the rod and cap].



C.5. Finite Element Modeling

Activating the finite element analysis module

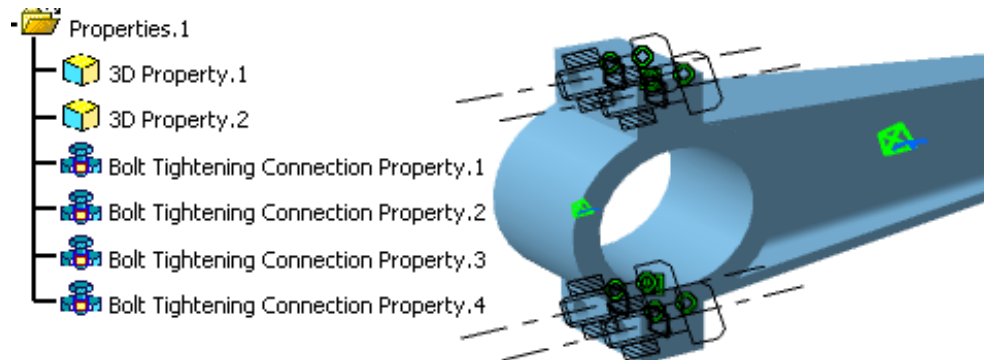
Start → Analysis & Simulation → Generative Structural Analysis → **New Analysis Case:** Static Analysis, ↵OK.

C.6. Modeling Geometric Constraints

Modeling bolt assembly connections



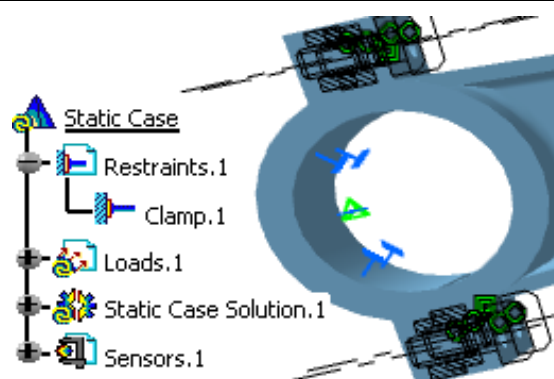
(Bolt Tightening Connection) → **Bolt Tightening Connection**: Supports [select the geometric restriction Coincidence.3 (Capac biela.1, Corp biela.1)], Tightening force: -100N, Orientation: ↓ Opposite [choose the direction so that the screw head area (on the symbol) is on the side acted upon by external force], ↵OK.
[Repeat this sequence for the other three Coincidence type geometric constraints].



Introducing displacement restrictions



(Clamp) → **Clamp**: Supports [select the inner semi-cylindrical face of the cap], ↵OK.

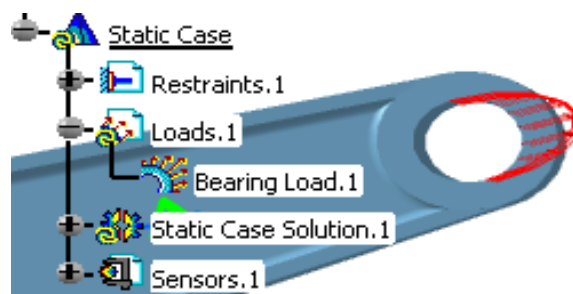


C.7. Modeling Loads

Introducing the force



(Bearing Load) → **Bearing Load**: Supports [select the cylindrical surface for the needle roller bearing seating]; Force vector X 3000N, Y 0N, Z 0N. Angle 180deg, Orientation Parallel; Profile Type Sinusoidal, ↵OK.



D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the calculation module



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

E. POST-PROCESSING OF RESULTS

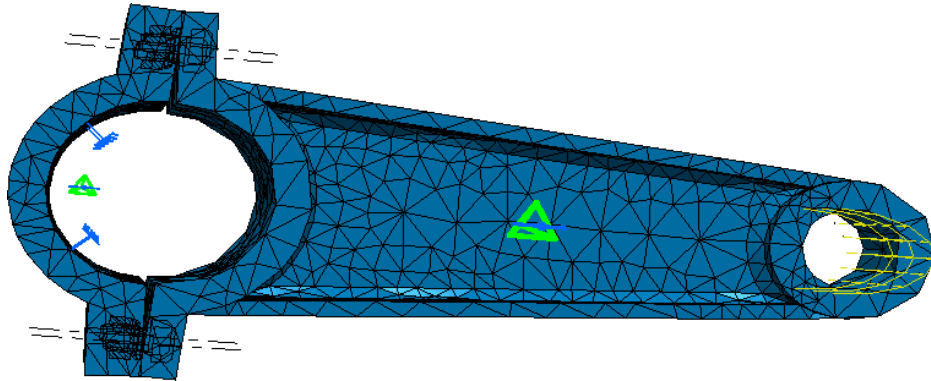
E.1. Visualizing the deformed state/Animation



(Deformation).



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



Observation

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

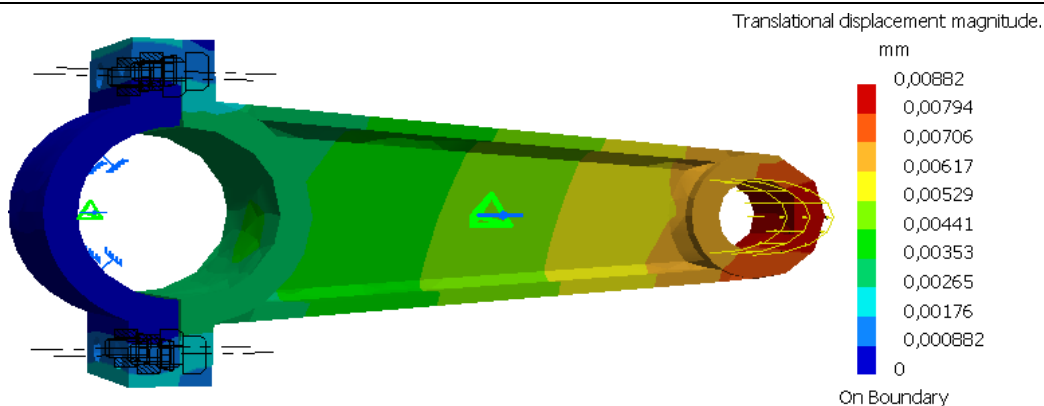


→ **Amplification magnitude**: Factor: 1 (other parameters can be set), ↵OK.

E.2. Visualizing the displacement field



(Displacement).

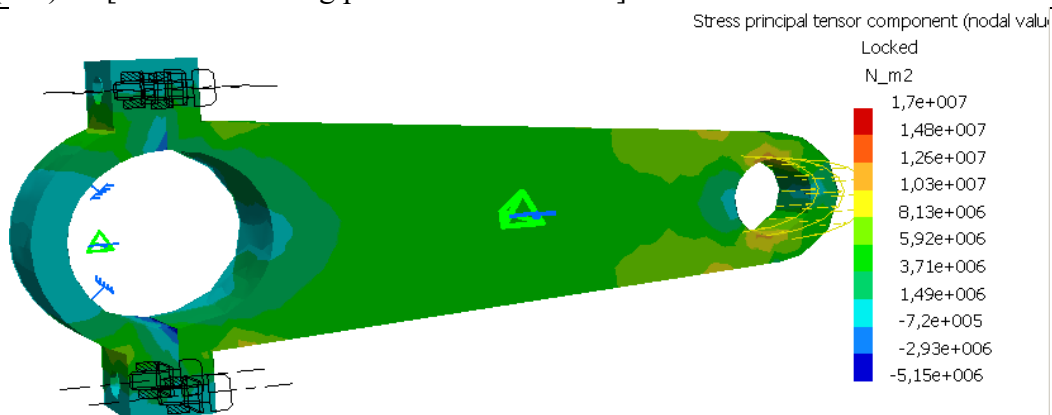


E.3. Visualizing the stress field

Visualizing the principal stress field in frontal section



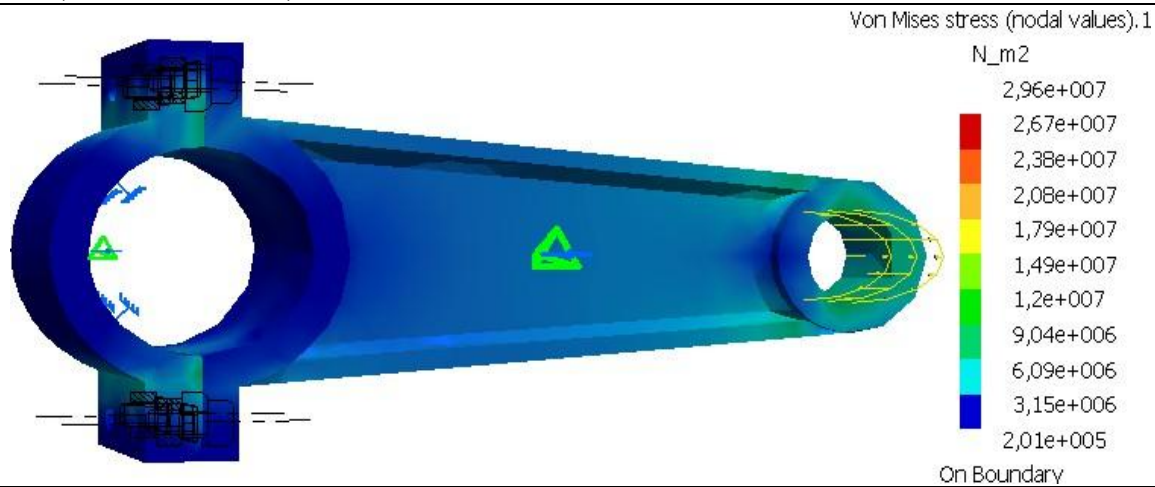
(Cut) → [orient the cutting plane with the mouse].



Visualizing the Von Mises stress field in frontal view



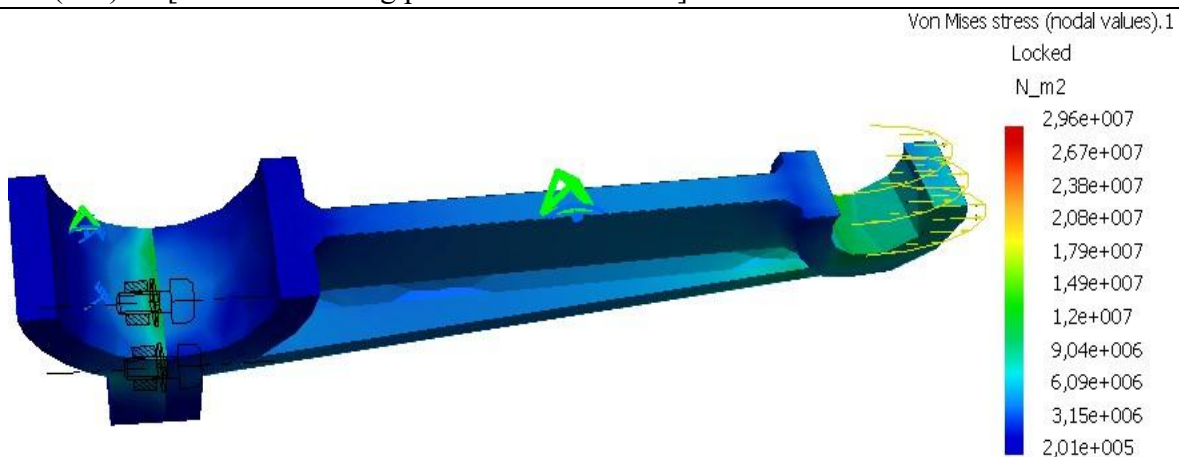
(Von Mises Stress).



Visualizing the Von Mises stress field in axial section



(Cut) → [orient the cutting plane with the mouse].



F. CONCLUSIONS

From the analysis of the displacement and stress fields, it is highlighted that the analyzed subassembly is a rigid structure (reduced maximum displacements – 0.00882 mm) with maximum stresses (29.6 MPa) in the load application area. It is observed that this value, for the OLC45 material (yield strength approx. 330 MPa), is acceptable, being smaller than the admissible limit. In case the maximum stress is larger or much smaller than the admissible limit, the structure is under-dimensioned or over-dimensioned, respectively, and dimensional modifications are required.

G. EXERCISE

Perform the finite element analysis algorithm from this application for the support lever subassembly presented in the following figure. This subassembly is mounted on a bar in a horizontal position via a bracket tightened by four M8 screws. The lever supports a weight $G = 3000 \text{ N}$ via a pin press-fitted into the lever bore and a tie rod.

The dimensions of the elements of the analyzed subassembly, presented above, are

$\phi_{1e} = 64 \text{ mm}$, $\phi_{1i} = 48 \text{ mm}$, $\phi_{2e} = 40 \text{ mm}$, $\phi_{2i} = 24 \text{ mm}$, $\phi_3 = 9 \text{ mm}$, $L = 260 \text{ mm}$, $l = 180 \text{ mm}$, $a = 104 \text{ mm}$, $b = 40 \text{ mm}$, $b_1 = 20 \text{ mm}$, $c = 8 \text{ mm}$, $g = 8 \text{ mm}$, $\gamma = 12$

mm. The inner edges of the body, lever, and bracket have fillets with radius $r = 2$ mm. The elements of the analyzed subassembly are made of steel, OL37.

