



FINITE ELEMENT METHOD

AEF-A.2.12

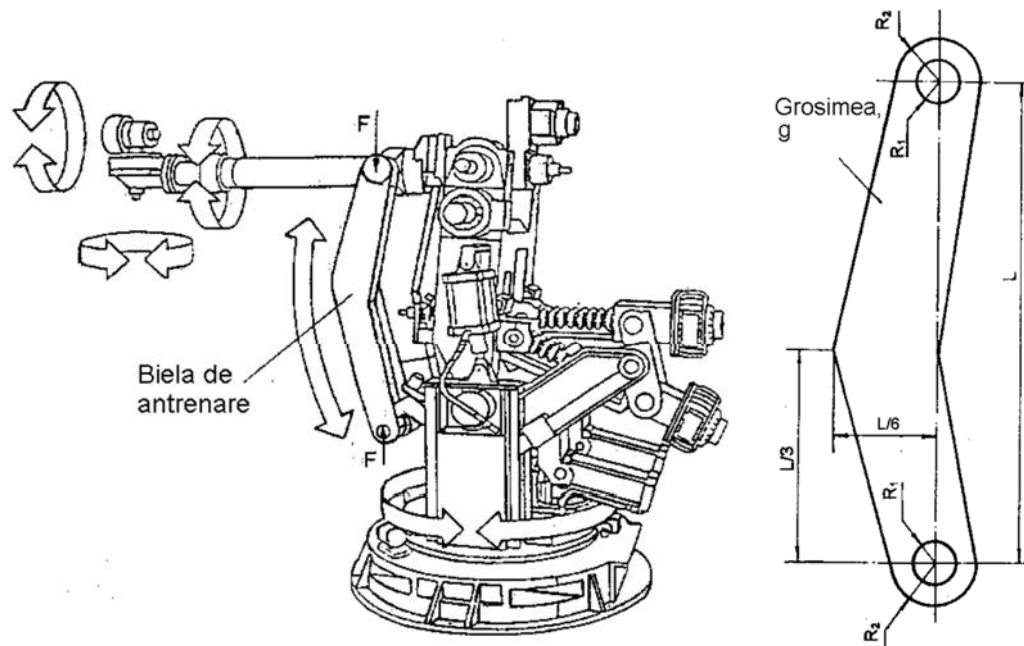
FLAMMING ANALYSIS USING FINITE ELEMENT METHOD OF BEAMS

INTRODUCTION

Mechanical systems are frequently encountered when, due to their shape (usually smaller parts in size than the other two), compression loads and joints, elements become unstable. The loss of stability of mechanical system elements can have catastrophic consequences on the entire system. The decision to study the elastic stability with FEM-based performance programs of the structural elements of the mechanical systems is usually taken by the design engineer who, as a preliminary analysis, understands the possibility of loss of stability.

DESCRIPTION OF THE APPLICATION

The drive lever of the industrial robot in the figure below, when the point of the effector element describes the workspace, may be required to compress. The shape and dimensions of the drive shaft made of aluminum alloy (duralumin) are shown in the figure below. The links of the drive shaft with the forearm and the drive element are made by poly-couples with needle radial bearings.



PURPOSE OF THE APPLICATION

In order to reveal the leakage of this lever by loss of stability, it is done the analysis with finite elements which aims to determine the critical forces for different forms of loss of stability of its structure. The dimensions of the drive shaft made of aluminum alloy are: $L = 900$ mm, $R_1 = 15$ mm, $R_2 = 25$ mm, $g = 12$ mm. The sharp edges of the front profile are connected with $R = 20$ mm.

Application AEF-A.2.12

BUCKLING ANALYSIS OF MECHANICAL BEAM STRUCTURES

A. DEFINITION OF THE APPLICATION

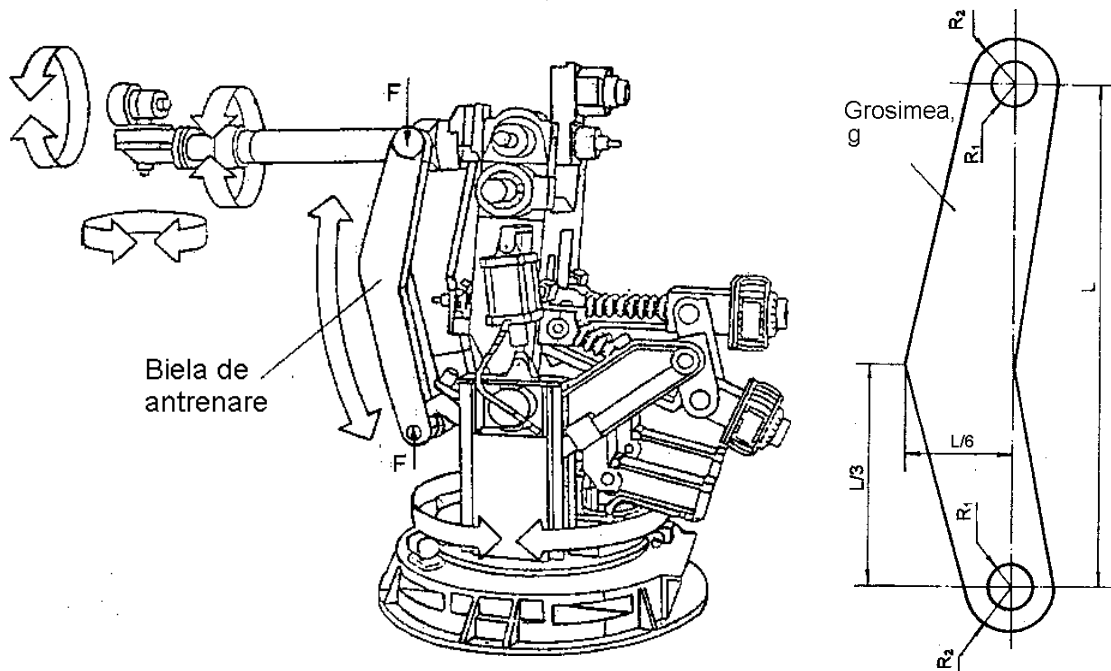
A.1. Introduction

In the practice of mechanical systems, situations are frequently encountered where, due to their shape (usually parts with one dimension much smaller than the other two), loads (compressive), and connections, elements become unstable. Failure through the loss of stability of mechanical system elements can have catastrophic consequences on the entire system.

The decision to study elastic stability using high-performance FEM-based programs for the structural elements of mechanical systems is usually taken by the design engineer who, following a preliminary analysis, foresees the possibility of stability loss.

A.2. Description of the application

The drive connecting rod of the industrial robot forearm in the figure below, when the characteristic point of the effector element describes the workspace, may be subjected to compression. The shape and dimensions of the drive connecting rod, made of aluminum alloy (duralumin), are presented in the figure below. The connections of the drive connecting rod with the forearm and the drive element are realized through multi-couples with radial needle bearings.



A.3. Purpose of the application

To highlight the failure of this lever through loss of stability, a finite element analysis is performed aiming to determine the critical forces for different eigenmodes of stability loss of its structure. The dimensions of the drive connecting rod, made of duralumin, are $L = 900$ mm, $R_1 = 15$ mm, $R_2 = 25$ mm, $g = 12$ mm. The sharp edges of the frontal profile are filleted with $R = 20$ mm.

B. PREPARATION OF THE ANALYSIS MODEL

B.1. Defining the analysis model

To prepare the finite element analysis model associated with the application above, the following must be identified:

- Geometric shape and dimensions.
- Restrictions induced by connections with adjacent elements.
- External loads.
- Material characteristics.

B.2. Description of the analysis model

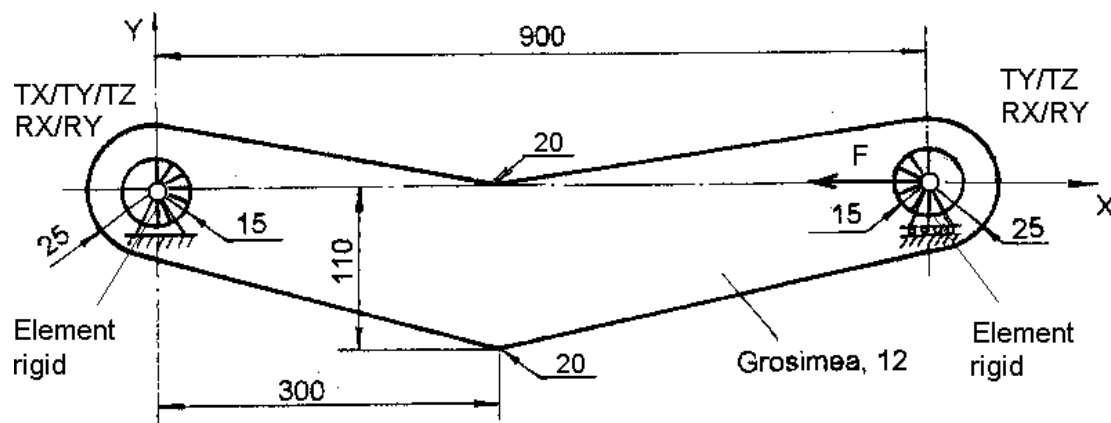
The figure below shows the geometric shape and dimensions associated with the analysis model. The element thickness is constant.

To achieve increased accuracy in simulating buckling behavior, considering the high stiffness of the bolts and the possibility of using idealized rigid elements, the bolts are replaced with a virtual rigid element equivalent to a system of radial bars with infinite stiffness connected to a central node.

The boundary conditions imposed on the model:

- Allow rotations around the axes of the considered virtual rigid elements.
- Take into account that the analyzed rod is guided laterally in the assembly areas with the connecting parts (rotations along X and Y axes in the rod plane are null).
- Cancel translations relative to the XYZ coordinate system for the master node of one rigid element by introducing a virtual rotation joint (**Pivot**).
- Cancel translations in the frontal plane (along OY and OZ axes) for the other master point.

To highlight the buckling direction, the model is loaded with a force $F = 1 \text{ N}$ in the direction of the bore centers (axis OX).



B.3. Establishing material characteristics

For the finite element analysis, the strength characteristics of the aluminum alloy (duralumin) from which the drive rod is made are:

- Longitudinal modulus of elasticity (Young's Modulus): $E = 73000 \text{ N/mm}^2$
- Transverse contraction coefficient (Poisson's Ratio): $\nu = 0,18$.

C. PRE-PROCESSING OF THE ANALYSIS MODEL

C.1. Geometric modeling

Activating the solid generation module and setting the unit of measurement

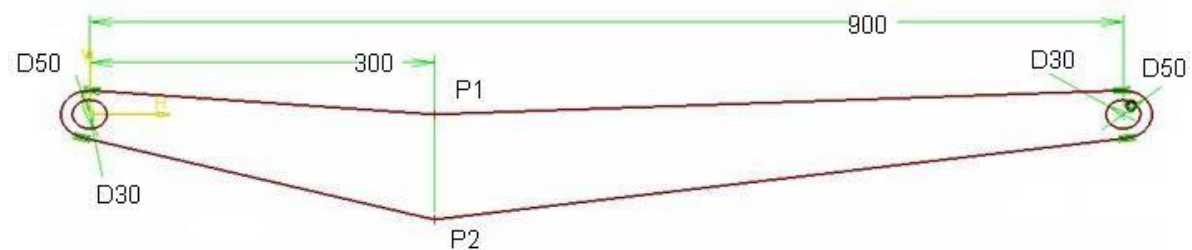
Start → Mechanical Design → Part Design → **New part**: New part name: Parghie.

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

↵OK.

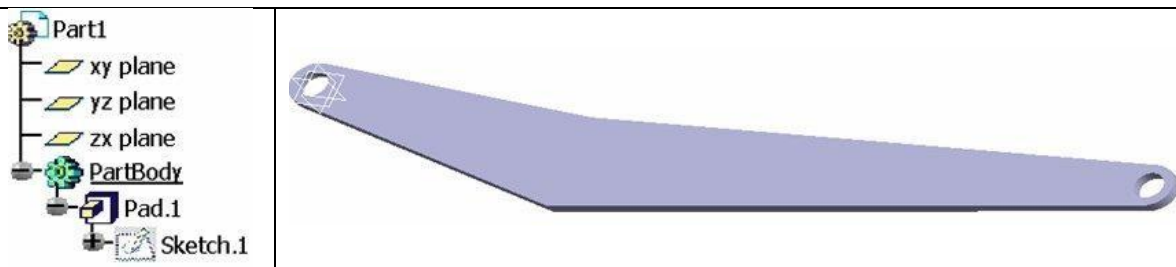
Generating the reference sketch

 (Sketcher) → xy plane →  (Profile) → [draw the closed contour of the rod section and the two holes] →  (Constraint) enter dimensions successively by selecting entities, using the **Ctrl** key for multiple selection] →  (Exit Workbench).



Generating the solid model

 (Pad) → **Pad Definition**: Length: 12, Selection: [select **Sketch.1** with the mouse], ↵OK.



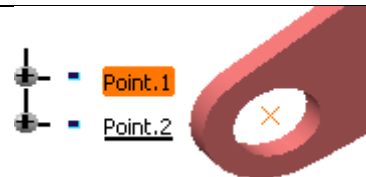
Generating fillets

 (Edge Fillet) → **Edge Fillet Definition**: Radius: 20 mm, Selection: [select the edges corresponding to point P1 using **Ctrl**], ↵OK. [Repeat this sequence for the fillet at point P2].



Generating points associated with hole centers

 (Point) → **Point Definition** Point type: Coordinates X 0mm, Y 0mm, Z 6mm Reference Point: Default (Origin) ↵OK. [Repeat this sequence for the point (900,0,6)].



C.2. Modeling material characteristics

[Select **PartBody** specification] →  (Apply Material) → **Library (ReadOnly)**: Metal, Aluminum, Apply Material; ↵OK. [Select **Aluminum** specification in the tree structure] → **Properties**: Feature Properties, Feature Name: DURAL; Analysis, Young Modulus 73000 N/mm², Poisson Ratio 0,18; Apply; ↵OK.

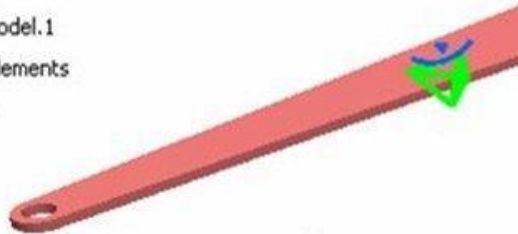
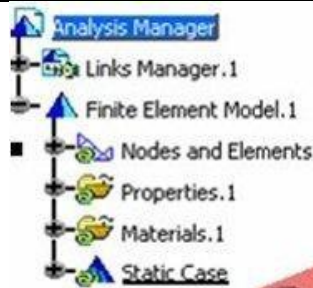


C.3. Finite element modeling

Activating the static finite element analysis module

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

[Select **OCTREE Tetrahedron Mesh**  in the tree structure or the symbol on the solid model structure] → **OCTREE Tetrahedron Mesh**: Size: 30 mm Absolute Sag: 8, Element type:  Parabolic .

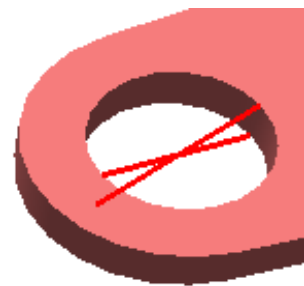
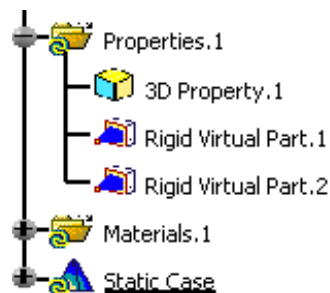


C.4. Modeling geometric constraints

Generating virtual rigid elements



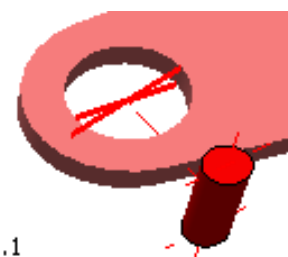
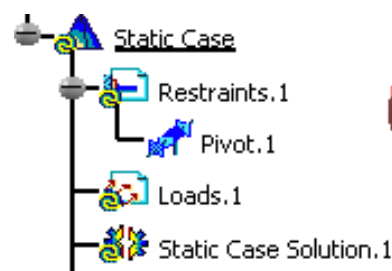
(Rigid Virtual Part) → **Rigid Virtual Part** → Supports: [select inner cylindrical surface of a hole]; Handler: [select the point inside the selected surface], . [Repeat this sequence for the other rigid element].



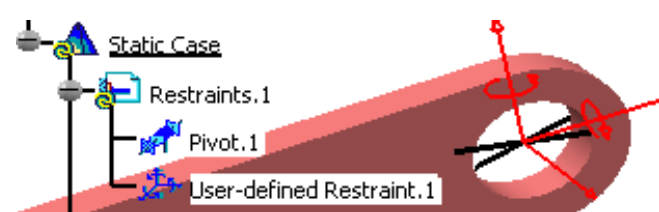
Introducing displacement restrictions



(Pivot) → **Pivot**: Supports: [select rigid element, **Rigid Virtual Part.1**], Released Direction: X 0, Y 0, Z 1, .



→ **User-defined Restraint**: Supports [select rigid element, **Rigid Virtual Part.2**]; [deactivate, ☐ Restrain Translation 1, ☐ Restrain Rotation 3], .

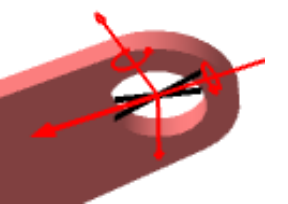
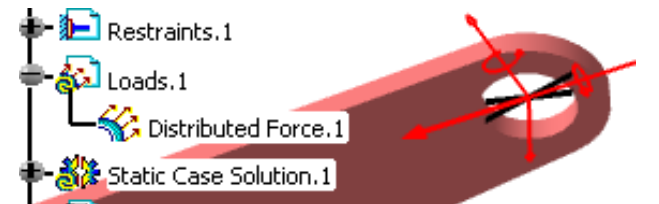


C.5. Modeling loads

Introducing the force



(Distributed Force) → **Distributed Force** [select element, **Rigid Virtual Part.2**]; Force vector X -1N, Y 0N, Z 0N. .



D. SOLVING THE FINITE ELEMENT MODEL

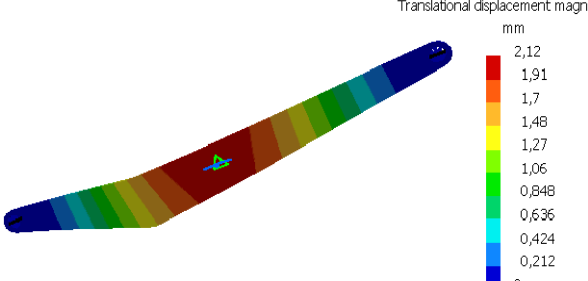
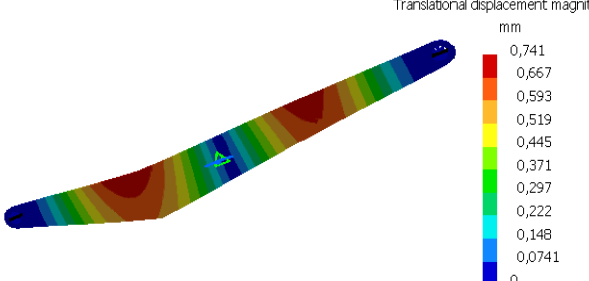
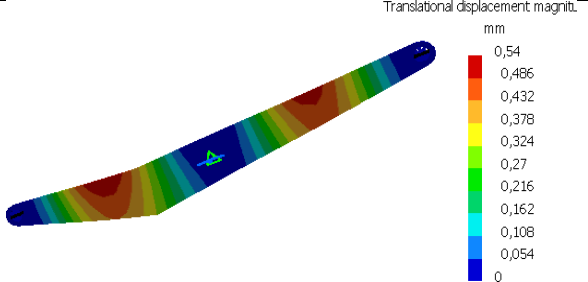
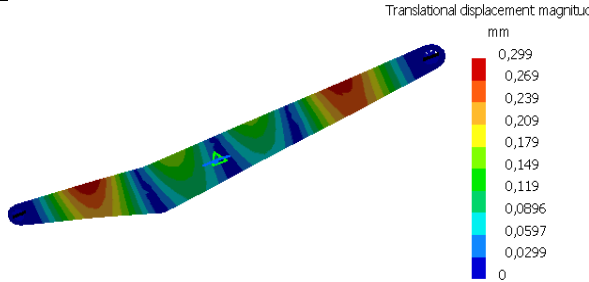
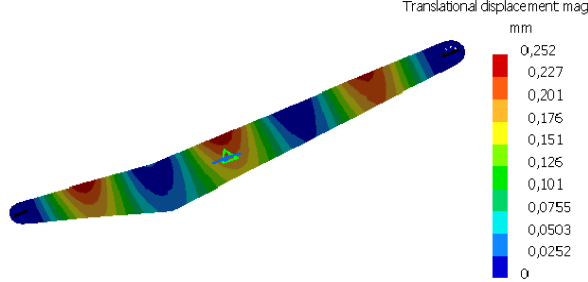
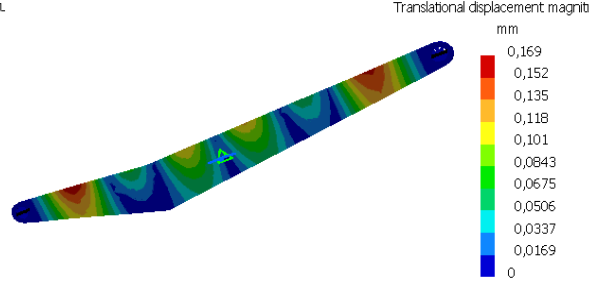
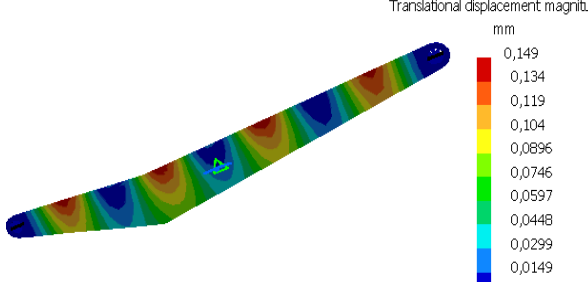
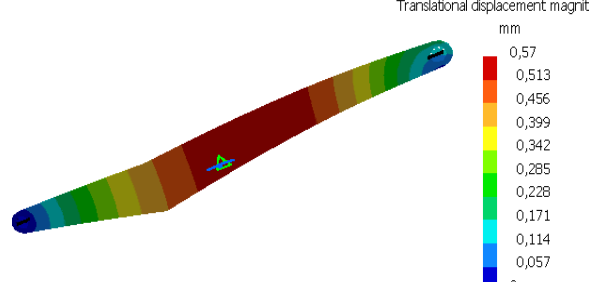
D.1. Launching the calculation module

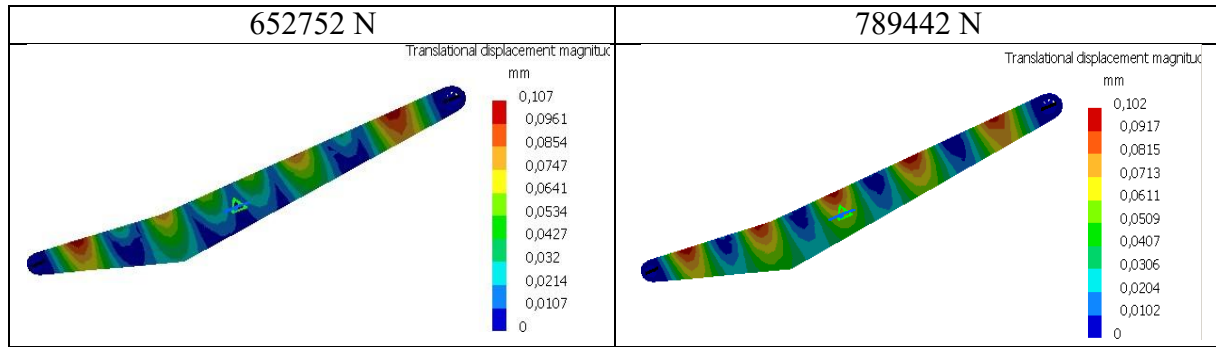


(Compute) → **Compute**:  All,  → **Computation Resources Estimation**,  → **Computation Status**.

Activating the finite element buckling analysis module	
Insert → Buckling Case → Buckling Case: Static Case Solution, Reference: Static Case Solution.1, Hide existing analysis case, ↵OK.	
D.3. Re-launching the calculation module	
 (Compute) → Compute: ↓ All, ↵OK → Computation Resources Estimation, ↵Yes → Computation Status.	

E. POST-PROCESSING OF RESULTS

E.1. Visualizing displacement fields / Buckling factors	
 (Displacement) → [right-click in the image area with displacements represented by vectors] → Translational displacement vector.1 object → Definition → Image Edition: Visu: Average iso; Occurrence: Buckling factor [select buckling factor value from the list], ↵OK.	
40178,6 N	83704,1 N
 Translational displacement magnitude mm 2,12 1,91 1,7 1,48 1,27 1,06 0,848 0,636 0,424 0,212 0	 Translational displacement magnitude mm 0,741 0,667 0,593 0,519 0,445 0,371 0,297 0,222 0,148 0,0741 0
155230 N	228271 N
 Translational displacement magnitude mm 0,54 0,486 0,432 0,378 0,324 0,27 0,216 0,162 0,108 0,054 0	 Translational displacement magnitude mm 0,299 0,269 0,239 0,209 0,179 0,149 0,119 0,0896 0,0597 0,0299 0
325309 N	420912 N
 Translational displacement magnitude mm 0,252 0,227 0,201 0,176 0,151 0,126 0,101 0,0755 0,0503 0,0252 0	 Translational displacement magnitude mm 0,169 0,152 0,135 0,118 0,101 0,0843 0,0675 0,0506 0,0337 0,0169 0
536911 N	545160 N
 Translational displacement magnitude mm 0,149 0,134 0,119 0,104 0,0896 0,0746 0,0597 0,0448 0,0299 0,0149 0	 Translational displacement magnitude mm 0,57 0,513 0,456 0,399 0,342 0,285 0,228 0,171 0,114 0,057 0



F. CONCLUSIONS

Determining critical buckling forces and eigenmodes of elastic stability loss for structural elements of mechanical systems using high-performance FEM-based programs is an important stage in designing their shape and dimensions. Finite element models, loads, and boundary conditions associated with them for elastic stability analysis have specificities related to the problem type and sometimes even the software used.

The critical buckling force associated with an eigenmode of stability loss is calculated with the relation $F_{ci} = F_i/F_0$, where F_i is the critical force factor associated with eigenmode i and F_0 is the concentrated force loading the analyzed structure. For the rod analyzed in this application, since the load $F = F_0 = 1N$, the critical buckling forces are equal to the associated buckling factors. The images above present the customized displacement field for the first 10 modes of stability loss associated with the respective critical buckling forces. If more stability loss forms are desired, the user can set the finite element analysis model for the desired number.

G. EXERCISE

Analyze with finite elements the stability loss if the connecting rod analyzed above is replaced with the one in the figure below. This is made of cast steel and has dimensions: $L = 180$ mm, $\phi_{1e} = 80$ mm, $\phi_{1i} = 50$ mm, $\phi_{2e} = 70$ mm, $\phi_{2i} = 40$ mm, $b = 40$ mm, $c = 7$ mm.

