Silviu Luis BUTNARIU Gheorghe Leonte MOGAN

Finite Element Analysis in mechanical engineering

-Practical applications in ANSYS-

Editura Universității Transilvania din Brașov

2024

EDITURA UNIVERSITĂȚII TRANSILVANIA DIN BRAȘOV

Adresa:	Str. Iuliu Maniu nr. 41A
	500091 Brașov
Tel.:	0268 476 050
Fax:	0268 476 051
E-mail:	editura@unitbv.ro

Editură recunoscută CNCSIS, cod 81

ISBN 978-606-19-1754-9 (ebook)

Lucrarea a fost avizată de Consiliul Departamentului de Autovehicule și Transporturi, Facultatea de Inginerie Mecanică a Universității Transilvania din Brașov.

Contents

Introduction

In an era marked by rapid innovation in engineering, numerical simulation has become an indispensable tool for optimizing projects and reducing costs. The Finite Element Method (FEM) has revolutionized the way engineers approach complex problems in industry.

As a powerful tool for engineering analysis, Finite Element Analysis (FEA) is used to solve problems from very simple to very complex. Due to time constraints and the limited availability of product information, it is mandatory to make some simplifications of the analysis models. This leads to the use of the FEM method by design engineers during the product development process. At the other end of the scale, specialized analysts implement FEA to solve very advanced problems such as vehicle crash dynamics, hydroforming, or airbag deployment.

ANSYS, one of the leading finite element analysis software, provides engineers with a powerful tool to analyze the behavior of complex structures and systems.

This collection of tutorials aims to guide mechanical and automotive engineering students in the effective use of ANSYS by providing practical examples and applications relevant to their fields of interest. Tutorials made in ANSYS offer a unique opportunity to gain hands-on experience in using industrial simulation software. Through the proposed exercises and projects, students will learn to model complex components and systems, apply realistic tasks and boundary conditions, and interpret the results obtained.

Authors

Application: FEA-A.1
Cantilever beam

KEY WORDS
Static linear analysis, Planar geometric model, Plane stress state, Linear material, Planar 2D finite element, Linear finite element, Machine element, Checking with classical models, Cantilever beam

CONTENT

A. PROBLEM DESCRIPTION

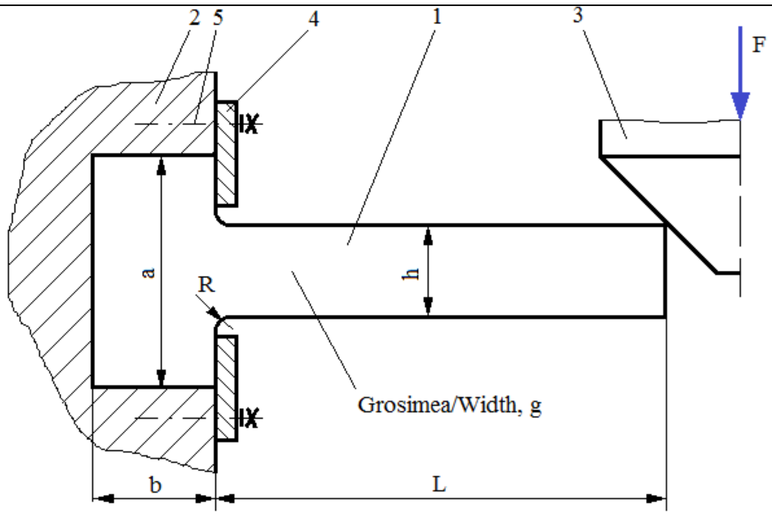
A.1 Introduction

In many training situations to solve FEA problems, especially as a beginner or initiating a new FEA platform, it is recommended to solve simple problems that are reduced to classic models with known analytical solutions.

The primary objective of this application involves developing an FEA for a rectangular beam bar structure and comparing the results with the classical analytical ones.

A.2 Application description

In the structure of the support device below, the elastic support member 1, firmly positioned in the body 2 by the rods 4 and the screws 5, must provide a displacement imposed by the push force F developed by the skate 3 and return to the state initially after its cancellation.



A.3 The application goal
For this application, it is necessary to analyse the displacement, deformation and tension fields of the bearing element 1 made of C55 steel and having the following dimensions: L = 100, h = 10 mm, g = 10 mm, a = 50 mm, b = 20 mm. After analysing the structure from the fact that the element 1 has a constant thickness and the force loading, F = 1000 N, it produces evenly the width, it is highlighted the framing of the problem in the flat tension state (the tensions are invariant to thickness).

B. THE FEA MODEL

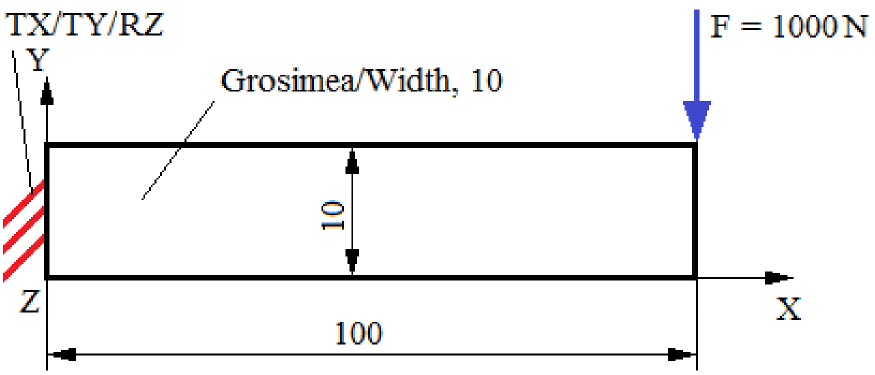
B.1 The model definition

In order to compare the results obtained by the finite element analysis with the classical solution model based on the material resistance methods (embedded beam), the most simplified possible model is adopted which implies:

- simple geometric shape,
- adoption of material strengthens constraints (embedding)
- the loads are concentrated,
- the material has a linear behavior

B.2 The analysis model description

Because the structure is framed in the FEA plane stress state, it can be modelled in plane, considering the rectangular geometric shape 100 mm long and 10 mm wide with 2D finite elements.
Geometric constraints involving cancellations of translation and rotation relative to the X, Y and Z axes, respectively, apply to the points on the Y-axis edge. Load the model with the concentrated force F = 1000 N in the far upper corner.



B.3 Choosing the characteristics of the material and the environment

For FEA, the strength characteristics of the C55 steel are:

- longitudinal modulus of elasticity, E = 210000 N / mm^2;
- Poisson's ratio, ν = 0,3.

Average working temperature of the subassembly, T_0 = 20 ° C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project

Unsaved Project - Workbench, Toolbox: ↵⊟ Analysis Systems : ↵↵ Static Structural (the window with project modules appears automatically); [change name, Static Structural].

Setting of problem type (2D)

↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options,

↵ Analysis Type, [selecting from drop down list ↵▼, ↵ 2D] → [close the window, ↵ X].

Saving of the project

↵ Save As... → WB Save As, File name: [enter name, FEA-A.1] → ↵ Save.

C.2 Modelling of material and environment characteristics

WB → Project Schematic → └ Engineering Data ✓ ◢ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [selecting from drop down list, C (Unit) with ↵▼, ↵ MPa], [enter in column, B (Unit) valoarea / value, 210000] → ↵ Update Project → ↵ Return to Project (others parameters are default).

C.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

WB → Project Schematic → └ DM Geometry → ↵ DM New Geometry... → ANSYS Workbench: ↵ ◉ Millimeter, ↵ OK

C.3.2 Sketch generation

Viewing default plane (XY)

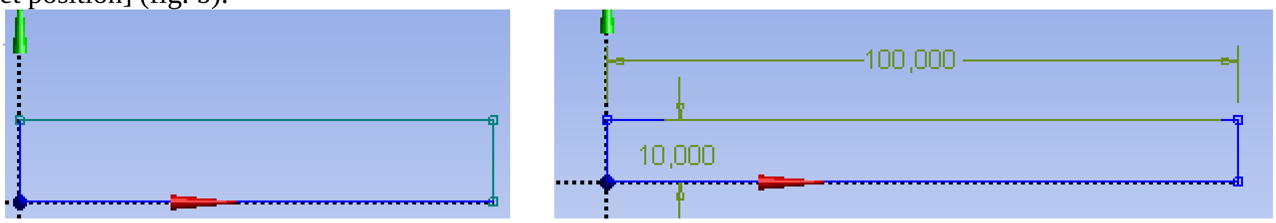
DM → ↵ Sketching → (Look At Face/Plane/Sketch) [automatically view of default plane, XY].

Rectangular line generation

↵ Draw → ↵ Rectangle → [trace rectangle line using pencil starting with, ↵, from coordinates system origin (appear symbol, P), and finish in opposite point simultaneously with release of the mouse fig. a)]

Dimensions

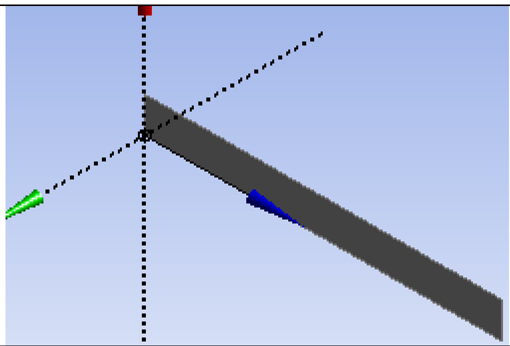
↵ Dimensions → ↵ Semi-Automatic → [automatically create dimensions with ↵] → Details View, ⊟ **Dimensions: 2**: ☐ L1, [enter value, 10]; ☐ L2, [enter value, 100] (fig. b). ↵ Display (view dimenions), Name: ↵☐ (deactivate), Value: ↵☑ (activate). ↵ Move (move dimensions), [activate dimension with, ↵ , and move kipping active until in target position] (fig. b).



a. ***b.***

C.3.3 Surface generation

DM → ↵ Concept → ↵ Surfaces From Sketches → Details View, ⊟ **Details of SurfaceSk1**: Base Objects → ↵ ⊞ ✓ XYPlane, ↵ ✓ Sketch1 → ↵ Apply; Thickness (>=0), [enter width value, 10] → ↵ Generate. └ ✓ Sketch1 → ↵ Hide Sketch (hide sketch). ↵ ISO (axonometric view).
↵ ⊞ ✓ 1 Part, 1 Body → ↵ ✓ Surface Body → Details View, ⊟ **Details of Surface Body**: Body, [enter name, *Suprafaţă bară*]



C.3.4 Save of geometric model

DM → ↵ (Save Project) → ↵ X (Close).

C.4. Finite element modelling

C4.1 Launching the finite element modelling module and setting the problem type, material characteristics, and unit system

Launching the modelling module with finite elements

WB → Project Schematic └ Model → ↵ M Edit... → [launching the *Mechanical ANSYS Multiphysics*].

Setting the type of the problem

→ Outline → **Project** → ↵ Geometry → Details of "Geometry", ⊟ **Definition** : 2D Behavior , [selecting from drop down list ↵, ↵ Plane Stress (default settings)].

Setting the material characteristics

Outline → ↵ ⊞ Geometry → ↵ Suprafață bară → Details of "Suprafață bară": **Material** : Assignment , [selecting from the list ↵, ↵ Structural Steel usually, when there is only one item, this setting is default).

Setting the units

→ ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) .

C.4.2 Model meshing

Automatic meshing (with implicit global parameters, including nonlinear finite element, parabolic with curved line side with an intermediate node)

→ Outline → **Project** → ↵ Mesh → ↵ Generate Mesh .

View mesh statistics

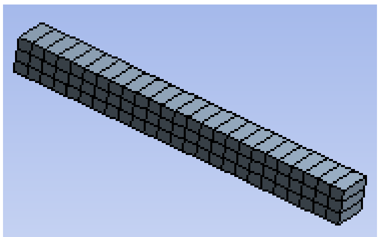
↵ Mesh → Details of "Mesh", ⊞ **Statistics** : Nodes , 282 ; Elements , 75 .

Adoption of the nonlinear finite element (with straight line or no intermediate node)

↵ Mesh → Details of "Mesh", ⊞ **Advanced** : Element Midside Nodes , [selecting from the list ↵, ↵ Dropped] → Update .

Revision of mesh statistics

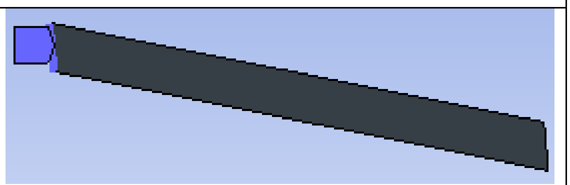
↵ Mesh → Details of "Mesh", ⊞ **Statistics** : Nodes , 104 ; Elements , 75 → ↵ ISO .



Obs. The last linear finite element mesh has the same number of finite elements (75) but has a smaller number of nodes (104) than the number of nodes (282) corresponding to the parabolic finite element mesh.

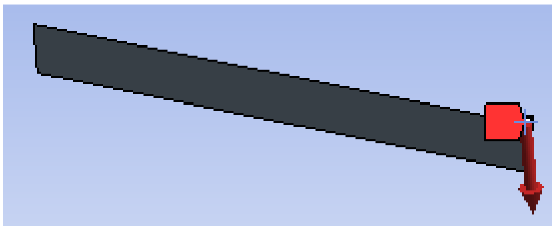
C.4.3 Supports and restraints modelling

→ Outline → ↵ ⊟ **Static Structural (A5)** → ↵ Supports → ↵ Fixed Support → Details of "Fixed Support", Geometry → ↵ (the line selection filter is activated) → [selecting with ↵ left edge] → ↵ Apply .



C.4.4 Loads modelling

→ Outline → ↵ ⊟ **Static Structural (A5)** → ↵ Loads → ↵ Force → Details of "Force", ⊟ **Scope** : Geometry → ↵ (the point selection filter is activated) → [selecting with ↵ the peak] → ↵ Apply ; ⊟ **Definition** : Define By , [selecting from drop down list ↵, ↵ Components], Y Component , ↵ 0, N (ramped) → [input balue, -1000].



C.4.5 Saving the project

→ ↵ File → Save Project... .

D. SOLVING THE FEA MODEL

D.1. Selecting the results types

→ Outline → ↵ Solution (A6) → ↵ Deformation → ↵ Total.

↵ Solution (A6) → ↵ Stress → ↵ Normal → Details of "Normal Stress", Definition: Orientation, [selecting from drop down list ↵, ↵ X Axis] (default selection).

↵ Solution (A6) → ↵ Stress → ↵ Shear.

↵ Solution (A6) → ↵ Stress → ↵ Equivalent (von-Mises).

D.2 Launching the solving module

→ ↵ Solution (A6) → ↵ Solve.

E. POST-PROCESSING OF RESULTS

E.1. Viewing the displacement field

→ Outline → ↵ Solution (A6) → ↵ Total Deformation.

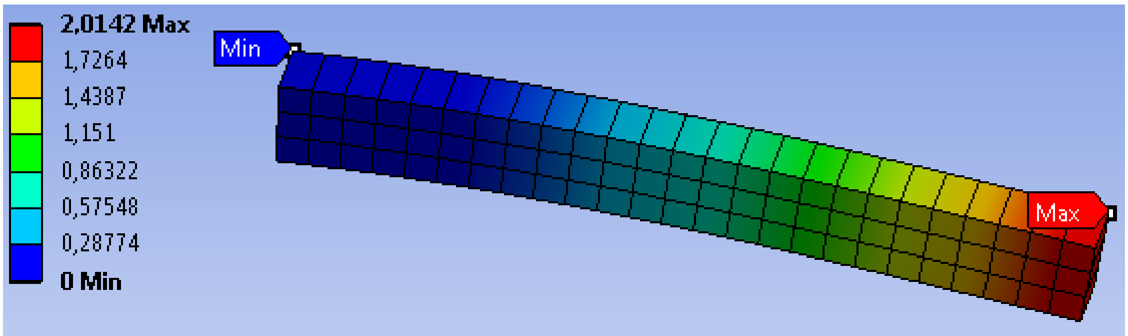
↵ → [selecting from drop down list ↵, ↵ Smooth Contours].

↵ → [selecting from drop down list ↵, ↵ Show Elements].

↵ Result → [selecting from drop down list ↵, ↵ 5, (2x Auto)].

Graph → ↵ Animation.

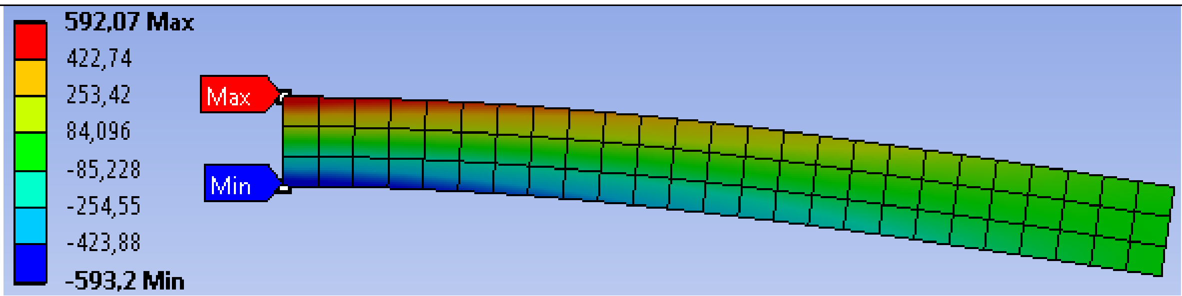
→ ↵ MAX; ↵ MIN.



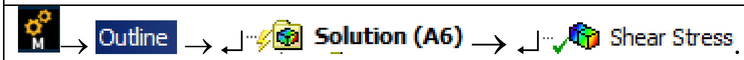
E.2. Viewing the stress fields

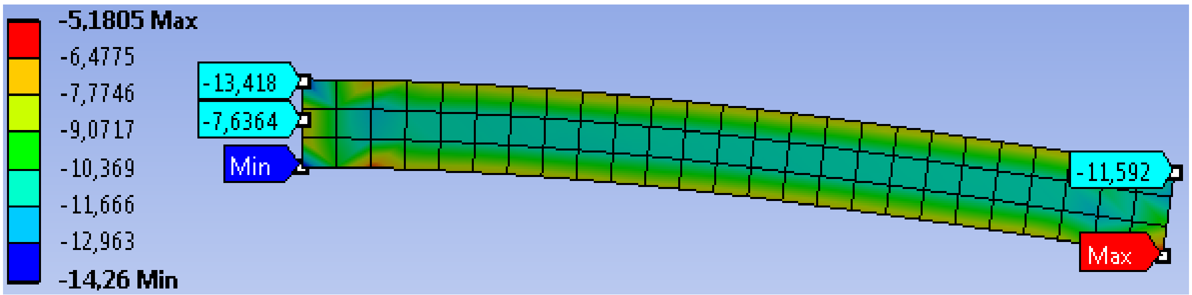
E.2.1 Viewing the normal stress in X direction

→ Outline → ↵ Solution (A6) → ↵ Normal Stress.

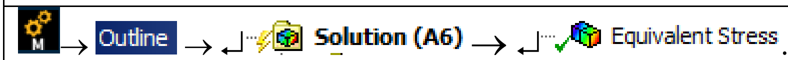


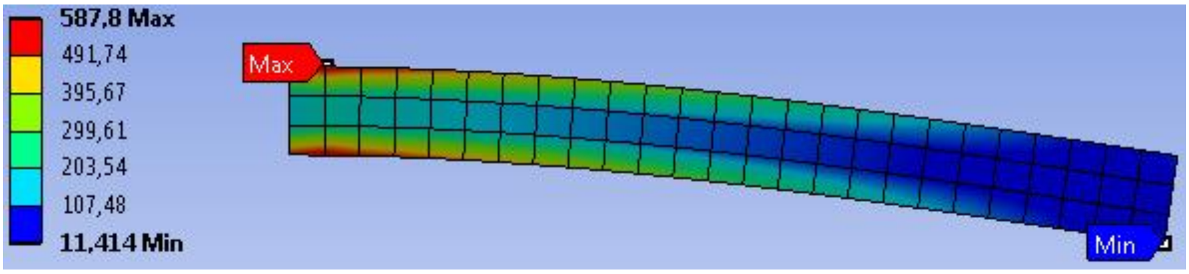
E.2.2 Viewing the tangential stress





E.2.3 Viewing the equivalent stress (von Mises)



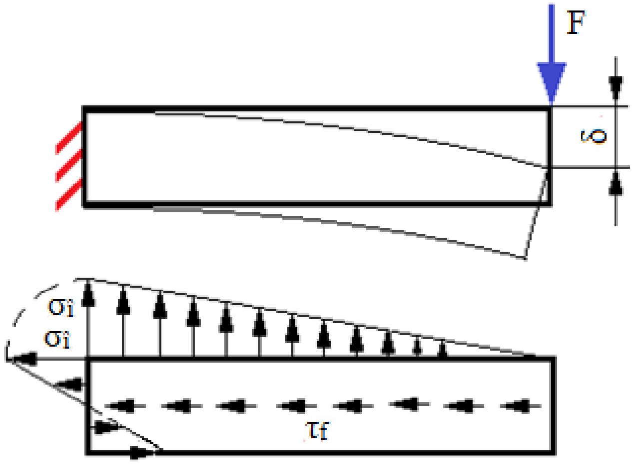


F. RESULTS ANALYSIS

F.1 Theoretical (analytical) calculus model

Classical analytical studies on the analysed structure (embedded bar) can be synthesized in the calculation of the parameters:

- maximum displacement,
    - $$\delta = \frac{Fl^3}{3\,EI_z} = \frac{4\,Fl^3}{E\,b\,h^3} = \frac{4\ 10^3\ 10^6}{2\ 10^5\ 10\ 10^3} = 2\ \text{mm};$$
- the maximum bending stress (Navier's relationship),
    - $$\sigma_{\hat{i}} = \frac{M_{\hat{i}}}{W_z} = \frac{6\,F\,l}{b\,h^2} = \frac{6 10^3\ 10^2}{10 10^2} = 600\ \text{MPa};$$
- maximum tangential shear stress (Juravschi's relationship),
    - $$\tau_f = \frac{3\ T}{2\ A} = \frac{3\ F}{2\,b\,h} = \frac{3 10^3}{2 10 10} = 15\ \text{MPa}.$$



F.2 Comparing and evaluation of the results
Taking into consideration the results obtained using the modelling and FEA (sub-chapter E) and the use of the classical calculus relations (subchapters E and F.1) obtained under the conditions of the strengthens of the materials, the following are highlighted: • the maximum displacement of 2,0142 mm obtained with FEA (E.1) is the same with the displacement (2 mm) obtained from the theoretical analytical model (sub-chapter F.1); • the maximum normal stress in the X-direction, 593,2 MPa, obtained with the finite element analysis (sub-chapter E.2.2) has a -1,13% deviation from the theoretical maximum normal stress (600 MPa) (sub-chapter F. 1); • the shear stress distribution (E.2.3) highlights maximum values (14.26 MPa) in the compressed clamp area having a deviation of - 4.9% from the theoretical value (15 MPa); • the equivalent stress (von Mises) has the maximum value (587.8 MPa) in the stretched clamp area.

G. CONCLUSIONS

On the first hand, the modelling and the finite element analysis from this application was done more with a teaching goal, in order to initiate the user with the main steps of developing an FEA application in ANSYS Workbench and, on the other hand, to compare and evaluate the results with some quasi-readings obtained through classical analytical models.

This process is recommended to be repeated for other practical situations in order to gain experience in developing analysis methods as well as evaluating the results.

The FEA model developed in this paper is inefficient from the point of view of the modelling possibilities offered by the ANSYS platform because it does not take into account the embedded connection area as well as the singularity associated with the concentric force due to the rough meshing with linear finite elements. These aspects are taken into account and studied in the application no. FEA-A.3.

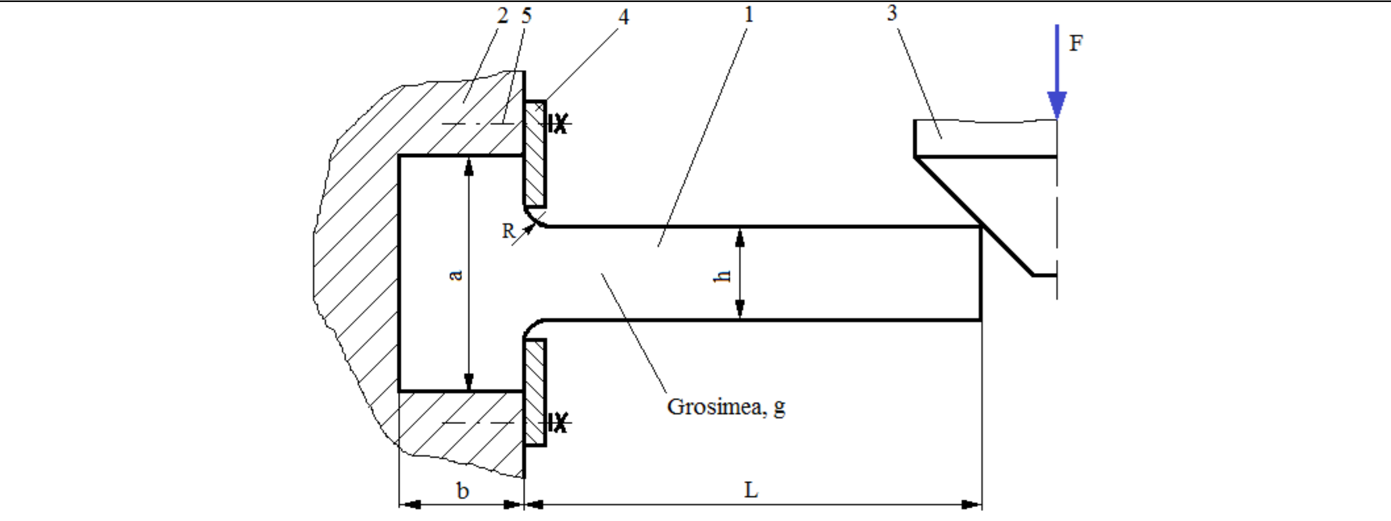
Application: FEA-A.2
Cantilever beam with fillet

KEY WORDS
Static linear analysis, Planar geometric model, Plane stress state, Linear material, Planar 2D finite element, Linear finite element, Parabolic finite element, Machine element, Checking with classical models, Cantilever beam, Singularities

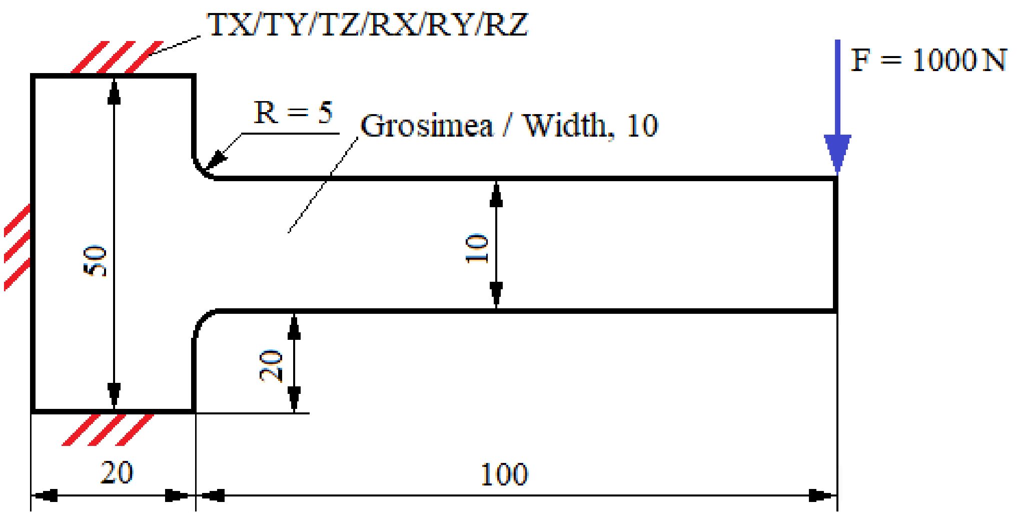
CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction
In many training situations to solve FEA problems, especially as a beginner or initiating a new FEA platform, it is recommended to solve simple problems that are reduced to classic models with known analytical solutions but also by highlighting the not recommended extreme situations (eg singularities) in the current practical applications. The main objective of this application involves the development of an FEA for a rectangular beam embedded on a contour, comparing the results with the classical analytical ones and highlighting the effects of the singularities involved by the theoretical cases of concentration of tensions in the zones of fillet with zero radius and in the zones of action of the concentrated forces on reduced surfaces at a point or a line.
A.2 Application description
In the structure of the support device below, the elastic support element 1, firmly positioned in the body 2 through the bars 4 and the screws 5, must ensure a displacement imposed under the action of the press force F, developed by the slide 3, and return to the initial state after its cancellation.
A.3 The application goal
In this application, the analysis of the displacement, deformation and tension fields of the support element 1 made of C55 steel and with the following dimensions is followed: L = 100, h = 10 mm, g = 10 mm, a = 50 mm, b = 20 mm . Starting from the fact that the element 1 has a constant constant thickness and the load with F = 1000 N is uniformly produced in width, it is emphasized that the problem is classified in the plane state of stresses (the voltages are invariable in thickness) and, therefore, the analysis with finite elements will be make customizations for this case. In addition, compared to the analysis from the application of FEA-A.1 the effects of the singularities of concentration of tensions in the "sharp" (null radius) and action areas of the "needle" (point) or "knife" (on the line) forces will be studied.



B. THE FEA MODEL

B.1 The model definition
In order to compare the results obtained by the finite element analysis with the classical solution model based on the material resistance methods (embedded beam), the most simplified possible model is adopted which implies: • simple geometric shape, • adoption of material strengthens constraints (embedding) • the loads are concentrated, • the material has a linear behavior
B.2 The analysis model description
Because the structure can be included in the FEA plane stress state, it can be modelled in plane, considering the rectangular geometric shape 100 mm long and 10 mm wide with 2D finite elements. Geometric constraints involving cancellations of translation and rotation relative to the X, Y and Z-axis, respectively, apply to the points on the Y-axis edge. Load the model with the concentrated force F = 1000 N in the far upper corner.

B.3 Choosing the characteristics of the material and the environment
For FEA, the strength characteristics of the C55 steel are: • longitudinal modulus of elasticity, E = 210000 N / mm^2; • Poisson's ratio, ν = 0,3. Average working temperature of the subassembly, T_0 = 20 ° C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project

Unsaved Project - Workbench: Toolbox: ↵ ⊟ Analysis Systems: ↵↵ Static Structural (the window with project modules appears automatically); [change name, Static Structural].

Setting of problem type (2D)

↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options, ↵ Analysis Type, [selecting from drop down list ↵▼, ↵2D] → [close the window ↵ ✕].

Saving of the project

↵ Save As... → Save As, File name: [enter name, FEA-A.2] → ↵ Save.

.2 Modelling of material and environment characteristics

→ Project Schematic → ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [selecting from drop down list C (Unit) cu / with ↵▼, ↵▼], [enter in column B (Unit) valoarea, 200000] → ↵ Update Project → ↵ Return to Project (others parameters are default).

C.3 Geometric modelling

C.3.1 *Model loading, DesignModeler (DM)*

→ Project Schematic → ↳ Geometry → ↵ New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵ OK.

C.3.2 *Sketch generation*

Viewing default plane (XY)

→ ↵ Sketching → (Look At Face/Plane/Sketch) [automatically view of default plane, XY].

Rectangular lines generation

↵Draw → ↵ Rectangle → [trace rectangle line using pencil starting with ↵ a point from left of Y axis, and finish in opposite point simultaneously with release of the mouse ↵] (fig. a) → [drawing two rectangular lines with the pencil indicator marking with, ↵ from a point of Y axis (C symbol appear), and finish in opposite point simultaneously with release of the mouse ↵] (fig. b).

Outline beam generation

↵Modify → ↵ Trim → [it will be deleted by selecting with ↵ the portions of the straight segments that do not belong to the contour (fig. c)].

Center lines in relation to the X axis

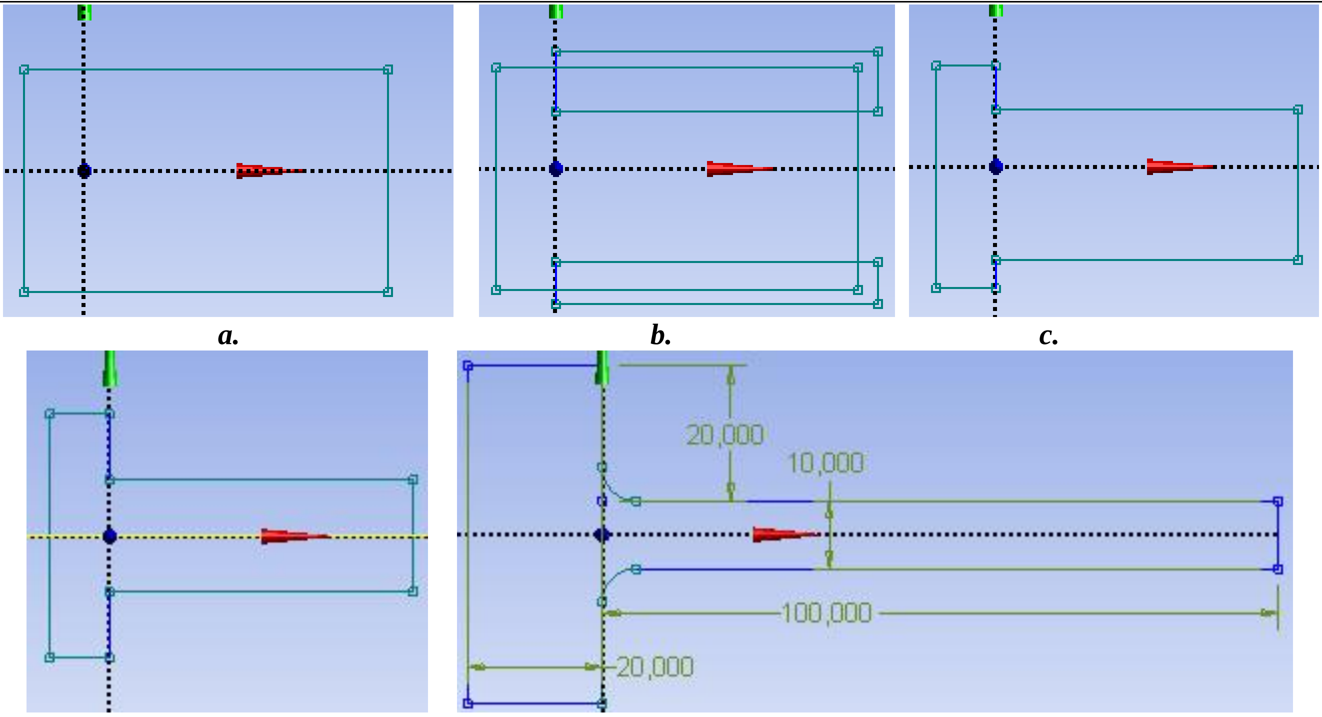
↵ Constraints →↵ Symmetry → [select with ↵ the X axis and then the two parallel lines with this axis to the left of the Y axis (fig. d)] → [select with ↵ the X axis and then the two parallel lines with this axis to the right of the Y axis (fig. d)].

Dimensions

↵Dimensions → ↵ Semi-Automatic → [dimensions are automatically activated with ↵] → Details View, ⊟ Dimensions: 4: → [they are inserted into the boxes ☐ L1, ☐ L2, ☐ L3, ☐ L4 (fig. e)]. ↵ Display (viewing dimensions), Name: ↵☐ (it is disabled), Value: ↵☑ (is activated). ↵ Move (moving dimensions), [the dimension activates with ↵ and moves keeping the activation to the desired position] (fig. e).

Fillet generation

↵Modify → ↵ Fillet → [input Radius:, radius value, 5] → [select with ↵ the connecting lines (fig. e)]

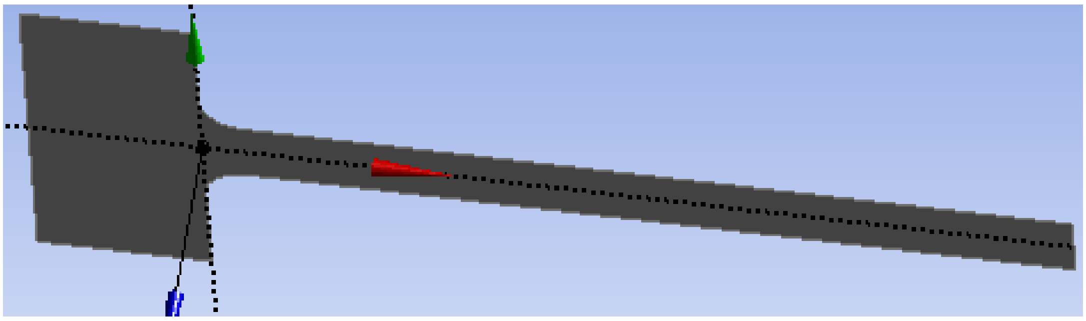


a. b. c.

d. e.

C.3.3 Surface generation

→ ↵Concept → ↵ Surfaces From Sketches → Details View, ⊟ **Details of SurfaceSk1**: Base Objects → ↵ Tree Outline → ⊞ XYPlane, ↵ Sketch1 → ↵ Apply; Thickness (>=0), [input thickness, 10] → ↵ Generate. ↳ Sketch1 → ↵ Hide Sketch (hide sketch) ↵ ISO (axonometric visualization).
↵ ⊞ 1 Part, 1 Body → ↵ Surface Body → Details View, ⊟ **Details of Surface Body**: Body, [input name, *Suprafață bară*].



C.3.4 Save of geometric model

→ ↵ (Save Project) → ↵ (Close Design Modeler).

C.4. Finite element modelling

C4.1 Launching the finite element modelling module and setting the problem type, material characteristics, and unit system

Launching the modelling module with finite elements

→ Project Schematic → ↳ Model → ↵ **Edit...** → [launching the *Mechanical [ANSYS Multiphysics*].

Setting the type of the problem

→ Outline → ↵ ⊞ Geometry → Details of "Geometry", ⊟ **Definition**: 2D Behavior, [select from list ↵, ↵ Plane Stress (default settings)].

Setting the material characteristics

Outline → ↵ ⊞ Geometry → ↵ Suprafață bară → Details of "Suprafață bară": **Material**: Assignment, [is selected from the list ↵, ↵ Structural Steel] (usually, when there is only one material, this setting is default).

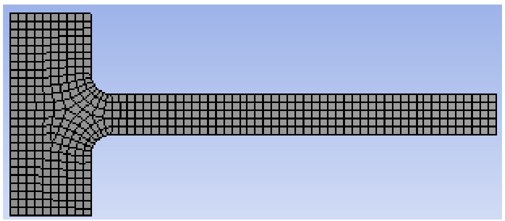
Setting the units

→ ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA).

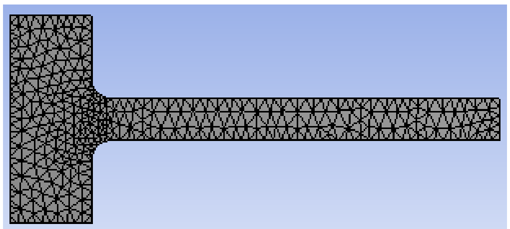
C.4.2 Model meshing

Setting general meshing features

→ Outline → ↵ Mesh → Details of "Mesh" → ⊞ Sizing : Use Advanced Size Function, [is selected from the list with ↵, ↵ Off]; Relevance Center, Use Advanced Size Function, [selecting from the list with ↵, ↵ Fine] ; ☐ Element Size, [enter the dimension of the finite element: 4, 2 (fig. a, b), 1, 0.5] (for each of the two cases below, 4 analyzes will be performed one for each value) → ⊞ Defaults : ☐ Relevance, [it is generated with ↵ , valoarea / value 100] → ⊞ Advanced : Element Midside Nodes, [selecting from the list ↵, ↵ Dropped/ Kept] (for each finite element dimension there will be 2 analyzes, one with linear finite element (without intermediate nodes on sides, variant Kept) and parabolic finite element with one intermediate node on sides, variant Kept).



a.



b.

Selecting triangular finit element (case I) or rectangular (case II)

↳ Mesh → Insert → Method → Details of "Automatic Method" - Method → ⊞ Scope : Geometry, [selecting with ↵ geometrical model] → ↵ No Selection, ↵ Apply → ⊞ Definition : Method, [selecting from the list ↵, ↵ Triangles/Quadrilateral Dominant].

Creating the model with finite elements

↵ Mesh → Generate Mesh (fig a,b).

View the meshing statistics

↵ Mesh → Details of "Mesh", ⊞ Statistics : ☐ Nodes, 422 / 605; ☐ Elements, 664 /516.

C.4.3 Supports and restraints modelling

→ Outline → ↵ Static Structural (A5) → ↵ Supports → ↵ Fixed Support → Details of "Fixed Support", Geometry → ↵ (activating line selection filter) → [selecting with Ctrl+↵ clamp edges] → ↵ Apply.

C.4.4 Loads modelling

→ Outline → ↵ Static Structural (A5) ↵ Loads → ↵ Force → Details of "Force", ⊞ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ peak] → ↵ Apply; ⊟ Definition : Define By, [selecting from the list ↵ , ↵ Components], ☐ Y Component, ↵ 0, N (ramped) → [input value -1000].

C.4.5 Saving the project

→ ↵ File → Save Project...

D. SOLVING THE FEA MODEL

D.1 Selecting the results

Selecting the total displacements

→ Outline → ↲ Solution (A6) → ↲ Deformation → ↲ Total.

Selecting the stress fields

↲ Solution (A6) → ↲ Stress → ↲ Equivalent (von-Mises).

Selecting the structural error

↲ Solution (A6) → ↲ Stress → ↲ Error.

Selection of the equivalent voltage field along a line from the connection base (fig. a)

Generation of Line 1: Model (A4) → ↲ Construction Geometry → ↲ Path → Details of "Path", Start: Start X Coordinate, [input value 5]; Start Y Coordinate, [input value, 5], (the coordinates of the point of upper fillet base, fig. a) → End X Coordinate, [input value, 5]; End Y Coordinate [input value, -5] (the coordinates of the point of lower fillet base, fig. b) .

Selecting the field of equivalent stress after Line 1:

↲ Solution (A6) → ↲ Stress → ↲ Equivalent (von-Mises) → Details of "Equivalent Stress 2" →

↲ Scope: ↲ Scoping Method, [selecting from the list ↲, ↲ Path]; Path, [selecting from the list ↲, ↲ Path].

Selecting the equivalent stress field after Line 1 (fig. b)

Generation of Line 2: Model (A4) → ↲ Construction Geometry → ↲ Path → Details of "Path", Start: Start X Coordinate, [input value 100]; Start Y Coordinate, [input value, 5], (the coordinates of the upper point, fig. b) → End X Coordinate, [input value, 100]; End Y Coordinate [input value, -5]; (coordinates of the lower point, fig. b).

Selecting the equivalent stress field after Line 2:

↲ Solution (A6) → ↲ Stress → ↲ Equivalent (von-Mises) → Details of "Equivalent Stress 3" →

↲ Scope: ↲ Scoping Method, [selecting from the list with ↲, ↲ Path]; Path, [selecting from the list with ↲, ↲ Path 2].

Selecting the field of structural error after Line 1 (fig. a)

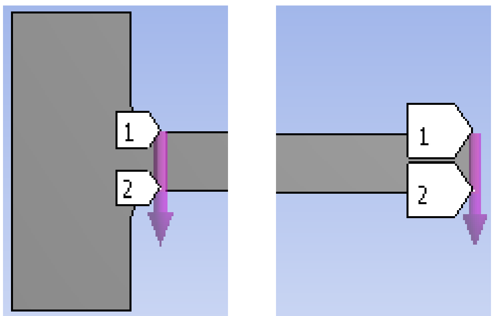
↲ Solution (A6) → ↲ Stress → Error → Details of "Structural Error" → ↲ Scope:

↲ Scoping Method, [selecting from the list ↲, ↲ Path]; Path, [selecting from the list ↲, ↲ Path].

Selecting the field of structural error after Line 2 (fig. b)

↲ Solution (A6) → ↲ Stress → Error →

Details of "Structural Error 2" → ↲ Scope: ↲ Scoping Method, [selecting from the list ↲, ↲ Path]; Path, [selecting from the list ↲, ↲ Path 2].



a. ***b.***

D.2 Launching the solving module

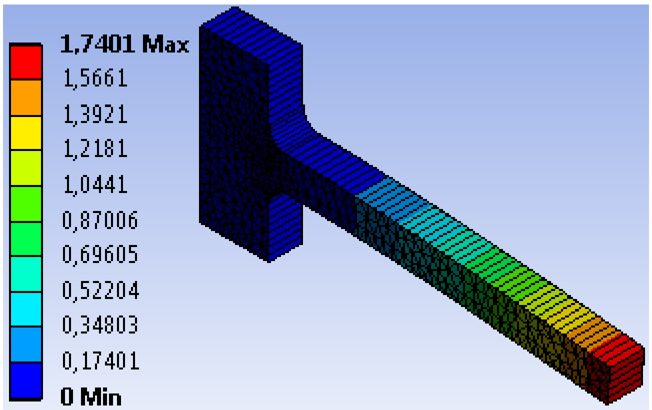
→ ↲ Solution (A6) → ↲ Solve.

E. POST-PROCESSING OF RESULTS

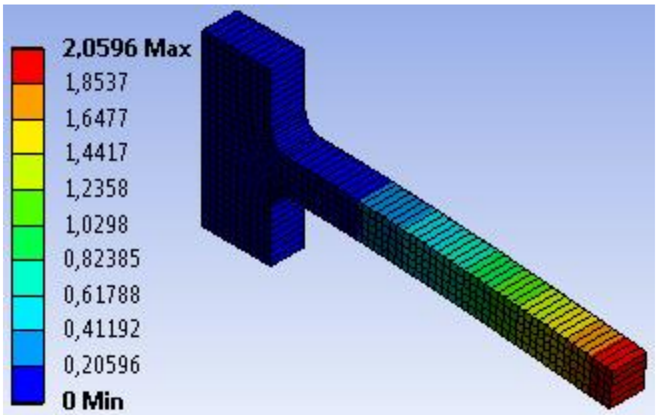
E.1 Viewing the total displacement field

→ Outline → ↵ Solution (A6) → ↵ Total Deformation (fig. a, for Case I fig. b, for Case II); ↵ ISO (axonometric visualization); ↵ → [selecting from the list ↵, ↵ Smooth Contours] (visualization of smooth contours) ; ↵ → [selecting from the list with ↵, ↵ Show Elements] (visualization of finite elements) ; ↵ Result → [selecting from the list with ↵, ↵ 3, (Auto Scale)] (selecting the scale of displacement); ↵ MAX (marking the node with the maximum total displacement); ↵ MIN (marking the node with the minimum total displacement).

Graph → ↵ Animation (visualisation af animation).



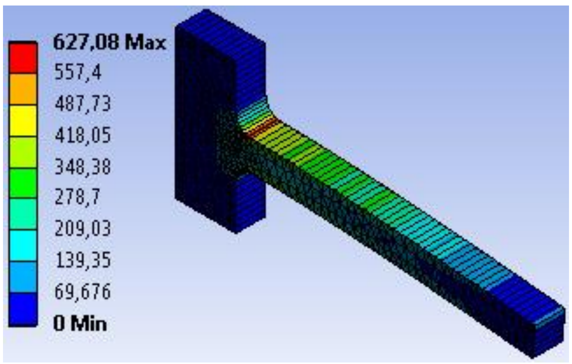
a.



b.

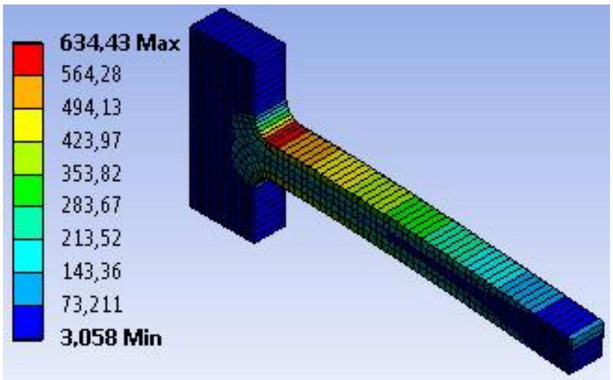
E.2 *Viewing equivalent stress fields*

→ Outline → ↵ Solution (A6) → ↵ Equivalent Stress (fig. a, for **case I**; fig. b, for **case II**);



a.

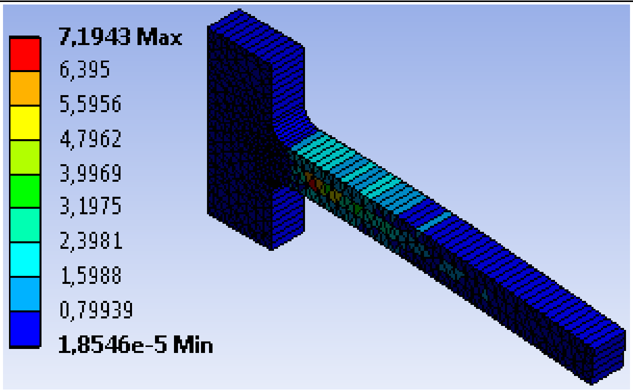
for



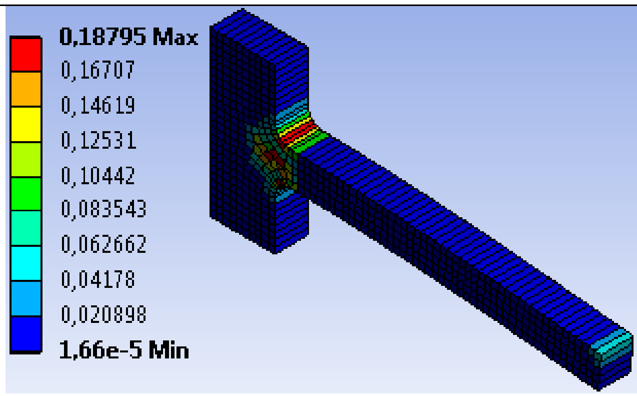
b.

E.3 *Viewing structural error fields*

→ Outline → ↵ Solution (A6) → ↵ Structural Error (fig. a, for **case I**; fig. b, for **case II**);



a.

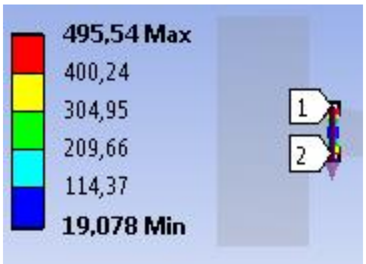


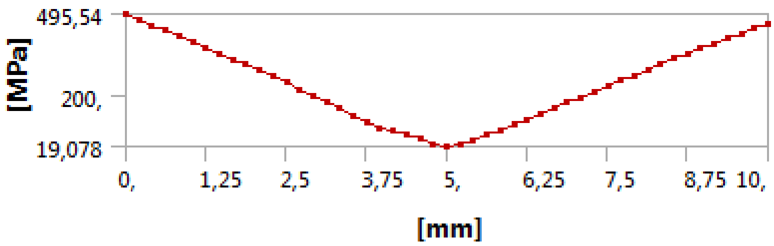
b.

E.3 *Viewing equivalent stress fields on line*

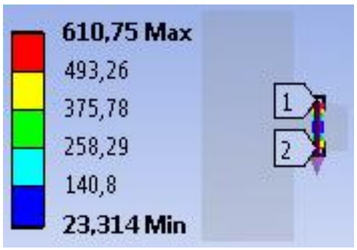
E.3.1 Viewing equivalent stress fields on Line1

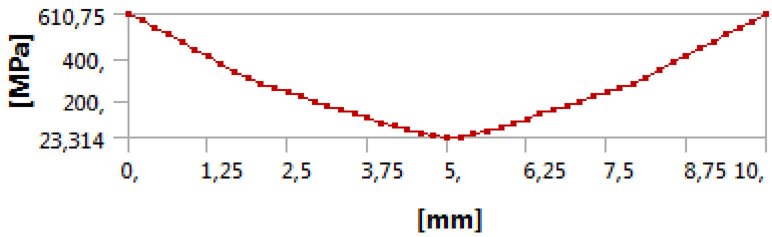
→ Outline → ↵ Solution (A6) → ↵ Equivalent Stress 2 (fig. a, for **case I**; fig. b, for **case II**);





a.

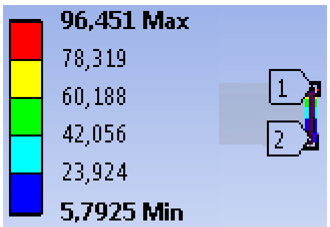


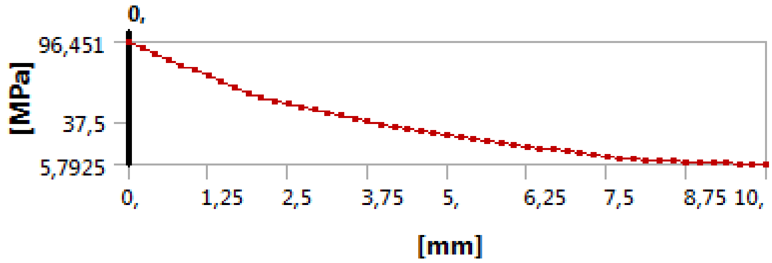


b.

E.3.2 Viewing equivalent stress fields on Line2

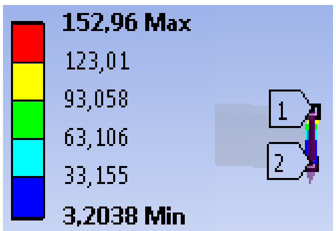
→ Outline → ↵ Solution (A6) → ↵ Equivalent Stress 3 (fig. a, for **case I**; fig. b, for **case II**);

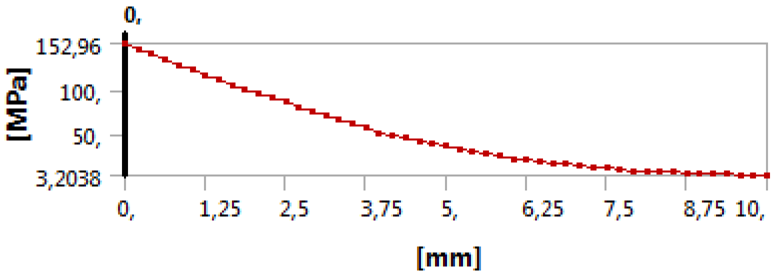




a.

a.



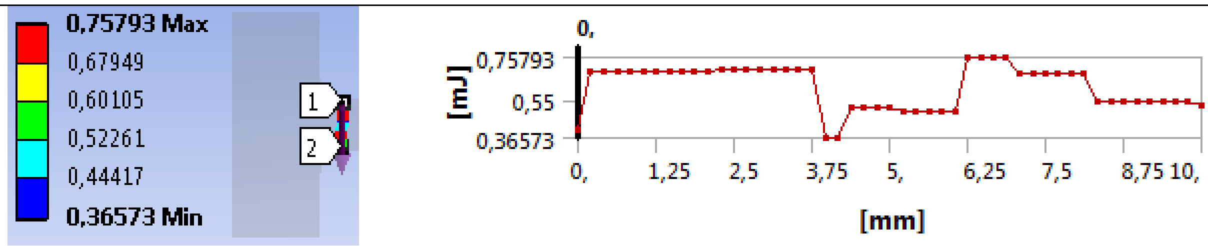


b

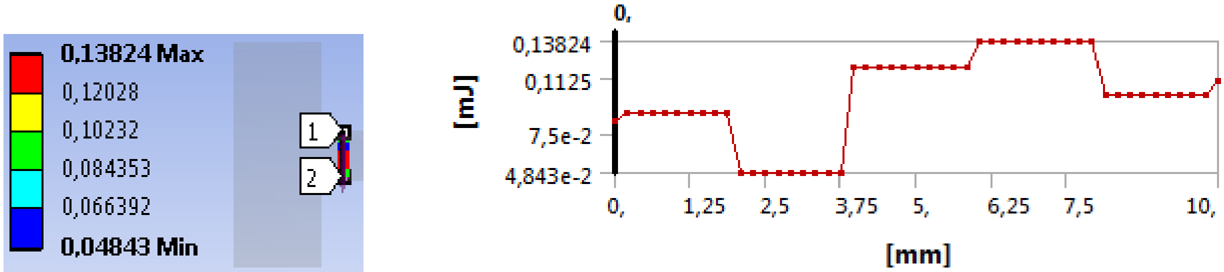
E.4 *Viewing structural error fields*

E.4.1 Viewing structural error fields on Line 1

Outline → Solution (A6) → Structural Error 2 (fig. a, for **case I**; fig. b, for **case II**);



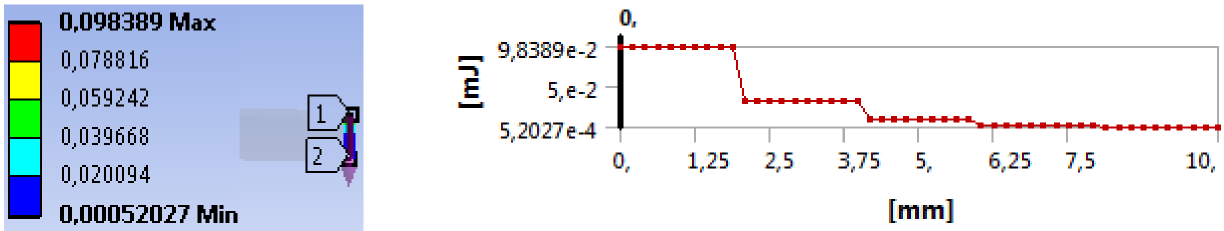
a.



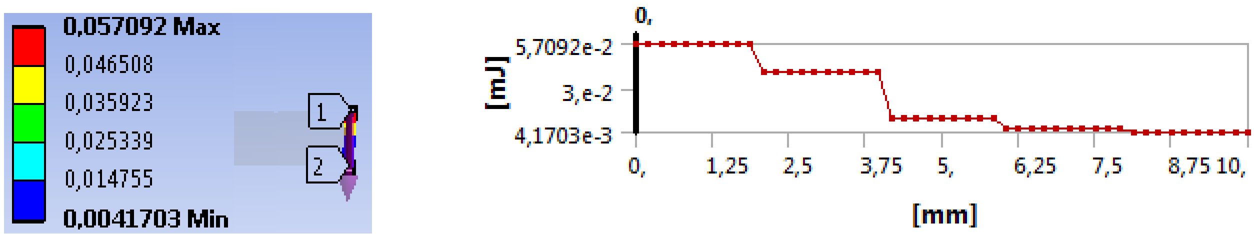
b.

E.4.2 Viewing structural error fields on Line 2

Outline → Solution (A6) → Structural Error 3 (fig. a, for **case I**; fig. b, for **case II**);



a.



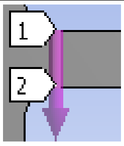
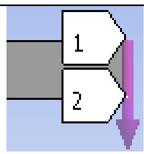
b.

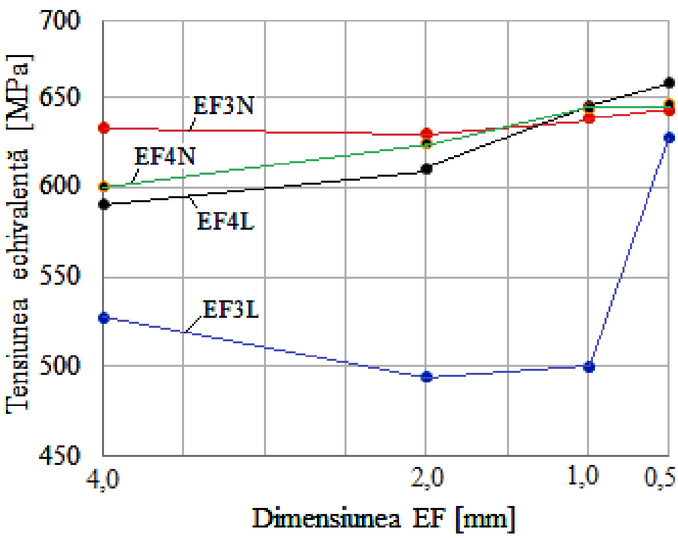
F. RESULTS ANALYSIS

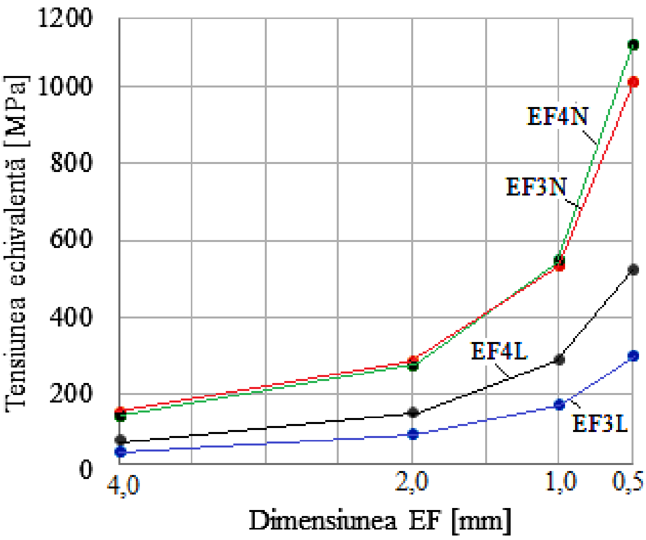
F.1 Summary of analysis results

In order to highlight aspects related to the accuracy of the results and the convergence of the solution, analyzes were performed considering for modeling linear finite elements (without intermediate nodes on sides) and nonlinear (with intermediate nodes on sides), with two forms (triangular and rectangular) and with four dimensions each. The results are summarized in the table and graphs below. In subchapter E, the results for the cases of the linear, triangular and rectangular finite element, with the dimension of 2 mm are presented (analyzes II and VI in the table).

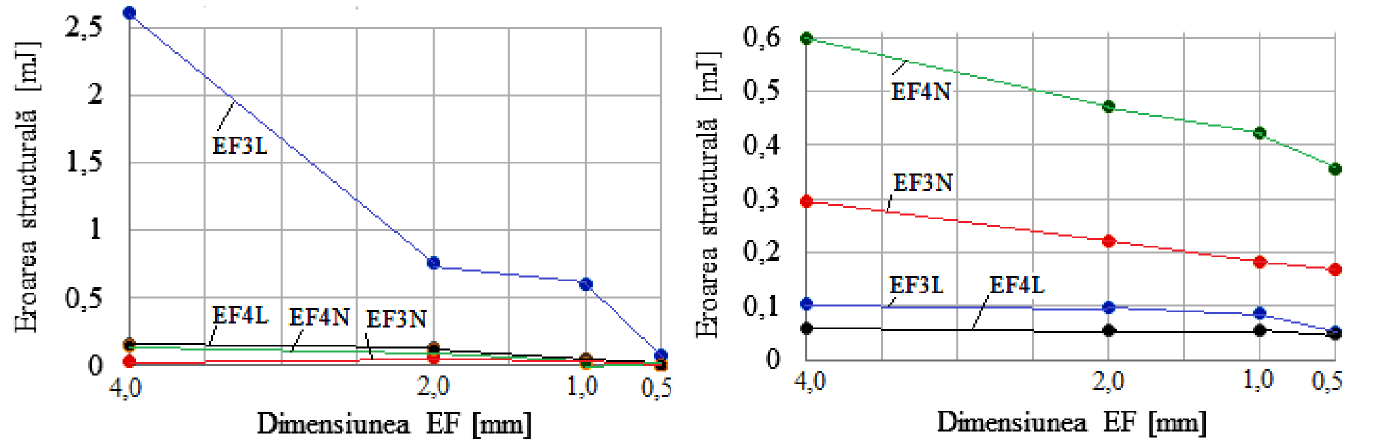
Tab. a

Analysis code	Dimension of finite element	Number of nodes	Number of finite elements	Total maximum displacements	 (Line 1)		 (Line 2)	
					Equivalent stress	Structural error	Equivalent stress	Structural error
First order triangular finit element (without intermediate nodes on edges, EL3L)								
I	4	194	284	1,447	527,48	2,619	53,26	0,105
II	2	422	664	1,74	495,54	0,758	96,451	0,098
III	1	1186	2034	1,958	500,08	0,606	176,53	0,087
IV	0,5	4210	774,6	2,026	628,65	0,081	302,03	0,052
Second order triangular finit element (parabolic, with intermediate nodes on edges, EL3N)								
V	4	671	284	2,011	633,91	0,0395	153,05	0,2951
VI	2	1607	664	2,024	629,91	0,0576	285,81	0,2233
VII	1	4405	2034	2,019	638,78	0,0437	537,15	0,1845
VIII	0,5	16165	7746	2,017	643,27	0,0013	1018	0,1695
First order rectangular finit element (without intermediate nodes on edges, EL4L)								
IX	4	230	179	2,054	591,14	0,1606	82,315	0,06
X	2	605	516	2,06	610,75	0,1383	152,96	0,057
XI	1	2169	2001	2,064	645,58	0,0485	291,52	0,055
XII	0,5	8382	8050	2,068	658,97	0,0059	525,04	0,047
Second order rectangular finit element (parabolic, with intermediate nodes on edges, EL4N)								
XIII	4	631	176	2,0366	600,91	0,14	145,61	0,6
XIV	2	1739	520	2,0383	624,74	0,125	275,72	0,4737
XV	1	6340	2001	2,036	644,81	0,0129	551,91	0,4225
XVI	0,5	24781	8038	2,036	646,81	0,0026	1111,14	0,356





a. ***b.***



c. *d.*

Following the analysis of the results obtained as a result of the modeling and FEA (subchapter E) and the use of the classical computational relations (the application A.1 subchapter F.1) obtained under the conditions of the materials strength hypotheses, the following are highlighted:

- The maximum total displacement for the case of the linear triangular finite element with large dimensions (4, 1 mm) has values (1.447 and 1.74 mm respectively according to lines I and II of the tables a and subchapter E.1, fig. A) with large deviations (-27.65%; -13%) from the value (2 mm) obtained from the theoretical analytical model (the application A.1 subchapter F.1).
- The maximum equivalent voltage in the connection area with voltage concentrator (visualized after a transverse line in the connection area) for the large triangular linear element (EF3l) with large dimensions (4, 2, 1 mm) has values (527.48; 495 , 54 and 500.08 MPa respectively according to lines I, II and III of the table, the graph of Fig. A and subchapter E.3.1, Fig. A) with large deviations (-18.84%; -28.37%; -23.06%) compared to the value of the maximum equivalent convergence voltage (approx. 650 MPa, fig. A); In the case of the other analyzes (cases I and II) - with large dimensions (4, 2 mm) we can see in fig. of much smaller deviations (<9%) of the values of the maximum equivalent voltage compared to the value of the maximum equivalent convergence voltage (approx. 650 MPa, fig. a).
- The maximum equivalent voltage in the head area of the bar (viewed from the force line) has small values (<300MPa, fig. B), for large dimensions (4, 2 mm) of the finite elements, and has much increased values (> 300MPa, fig. B); the value of the equivalent voltage increases exponentially (fig. b) with the increase of the discretization fineness (EF dimensions <0.5 mm) which highlights the singularity effect of the tension consequence of the load with concentrated force - a theoretical situation in which much increased values of the voltage result around the force action point (tensions in this area are not taken into consideration when designing).

F.2 Analysis of convergence and precision

Following the analysis of the tensile values of the structural errors (tab. A, fig. C, d) the following are highlighted:

- In fig. c, table. a, but also in subchapter. E.4.1, fig. to highlight the variation of the structural error with the increase of the discretization fineness. The increased values of the structural error (> 0.5 mJ) for the linear triangular finite element (EF3L) with enlarged dimensions (> 1 mm) in correlation with the variation of the equivalent voltage in fig. to also highlight in this way the increased deviations of the voltages from the quasi-real value (approx. 650 MPa, fig. a). For the other finite elements (EF4L, EF4N and EF3N) we observe reduced values of structural error (<0.16 mJ) and for large finite element dimensions (4, 2 mm) that decrease, in correlation with the increase (convergence) of the stresses. equivalent (fig. a), with the increase of the discretization fineness (decrease of the finite element size).

- In fig. d we observe reduced values of the structural error (<0.6 mJ, for EF3N) and their decreasing variations, which is not correlated with the exponential increase of the maximum voltages (fig. b), thus highlighting the singularity (non-convergence) of the voltage. .
- In fig. to highlight the quasi-exact equivalent voltage (approx. 650 MPa, fig. a) as a result of the convergence of the solution (asymptotically approximating a quasi-exact value for finite elements of different shapes as the fineness of discretization increases (EF dimension decreases).

G. CONCLUSIONS

The modeling and analysis with finite elements of this paper were done more with didactic purpose aiming, on the one hand, the initiation of the user with the main stages of developing an application of FEA in ANSYS Workbench and, on the other hand, the comparison. and evaluating the results obtained from FEA with different shapes and sizes of finite elements.

The adopted FEA model leads to coarse deviations from the exact solution for the linear triangular finite element as opposed to the quadratic finite element model which shows a convergence with very small deviations.

The analysis of the results, in particular, of the tensions, for discretizations with increased fineness, shows that in the area with the singularity of the tension (the point of application of the concentrated force), although the structural error decreases to allowable values which would show a good accuracy, the values of the tensions do not converge towards the cvsiexact value, but they grow non-asymptotically (values that do not correspond in reality).

The FEA model studied in this paper is efficient in terms of modeling possibilities offered by the ANSYS platform, especially for the quadrilateral finite element that ensures good convergence in the connection area (voltage concentrator, real case) and leads to increased errors. , non-convergence of the solution for the force action area is concentrated in one point (theoretical case), a case not recommended in the design practice that can be avoided by considering the force distributed on a line very close to reality.

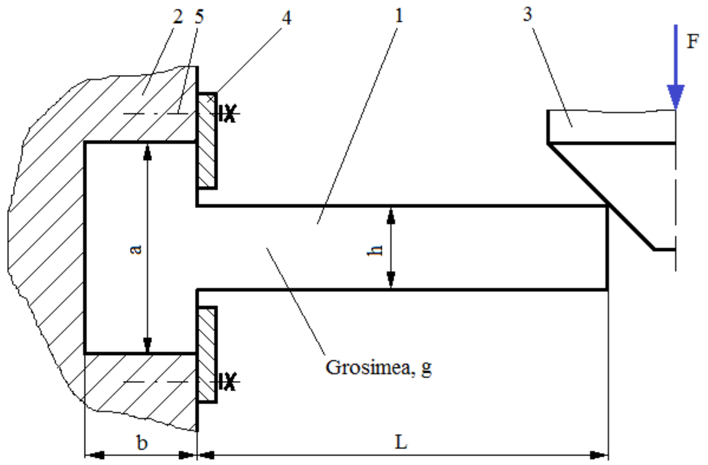
Application: FEA-A.3
Cantilever beam with singularities

KEY WORDS
Static linear analysis, Planar geometric model, Plane stress state, Linear material, Planar 2D finite element, Linear finite element, Parabolic finite element, Machine element, Checking with classical models, Cantilever beam, Singularities

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction
In many training situations to solve FEA problems, especially as a beginner or initiating a new FEA platform, it is recommended to solve simple problems that are reduced to classic models with known analytical solutions but also by highlighting the not recommended extreme situations (e.g. singularities) in the current practical applications. The main objective of this application involves the development of an FEA for a rectangular beam embedded on a contour, comparing the results with the classical analytical ones and highlighting the effects of the singularities involved by the theoretical cases of concentration of stresss in the zones of fillet with zero radius and in the zones of action of the concentrated forces on reduced surfaces at a point or a line.
A.2 Application description
In the structure of the support device below, the elastic support element 1, firmly positioned in the body 2 through the bars 4 and the screws 5, must ensure a displacement imposed under the action of the press force F, developed by the slide 3, and return to the initial state after its cancellation.
A.3 The application goal
In this application, the analysis of the displacement, deformation and stress fields of the support element 1 made of C55 steel and with the following dimensions is followed: L = 100, h = 10 mm, g = 10 mm, a = 50 mm, b = 20 mm . Starting from the fact that the element 1 has a constant constant thickness and the load with F = 1000 N is uniformly produced in width, it is emphasized that the problem is classified in the plane state of stresses (the stresss are invariable in thickness) and, therefore, the analysis with finite elements will be make customizations for this case. In addition, compared to the analysis from the application of FEA-A2 the effects of the singularities of concentration of stresss in the "sharp" (null radius) and action areas of the "needle" (point) or "knife" (on the line) forces will be studied.



B. THE FEA MODEL

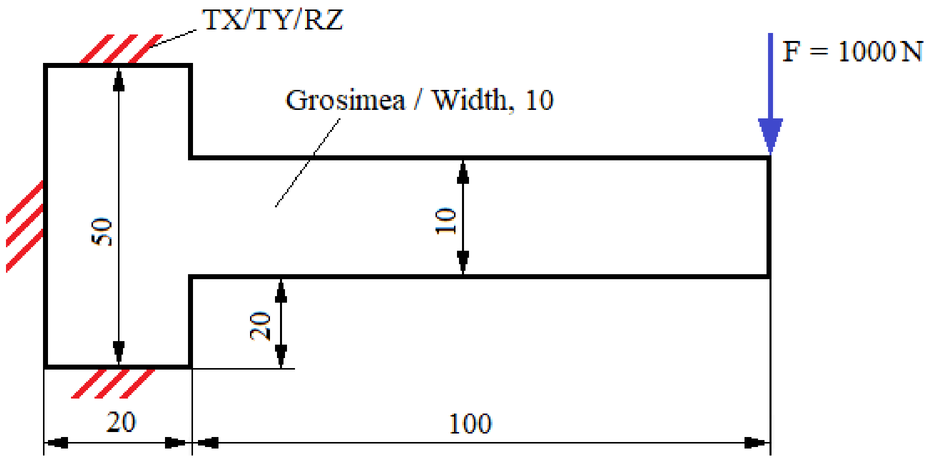
B.1 The model definition

In order to compare the results obtained by the finite element analysis with the classical solution model based on the material resistance methods (embedded beam), the most simplified possible model is adopted which implies:

- simple geometric shape,
- adoption of material strengthens constraints (embedding)
- the loads are concentrated,
- the material has a linear behaviour

B.2 The analysis model description

Because the structure can be included in the FEA plane stress state, it can be modelled in plane, considering the rectangular geometric shape 100 mm long and 10 mm wide with 2D finite elements.
Geometric constraints involving cancellations of translation and rotation relative to the X, Y and Z-axes, respectively, apply to the points on the Y-axis edge. Load the model with the concentrated force F = 1000 N in the far upper corner.



B.3 Choosing the characteristics of the material and the environment

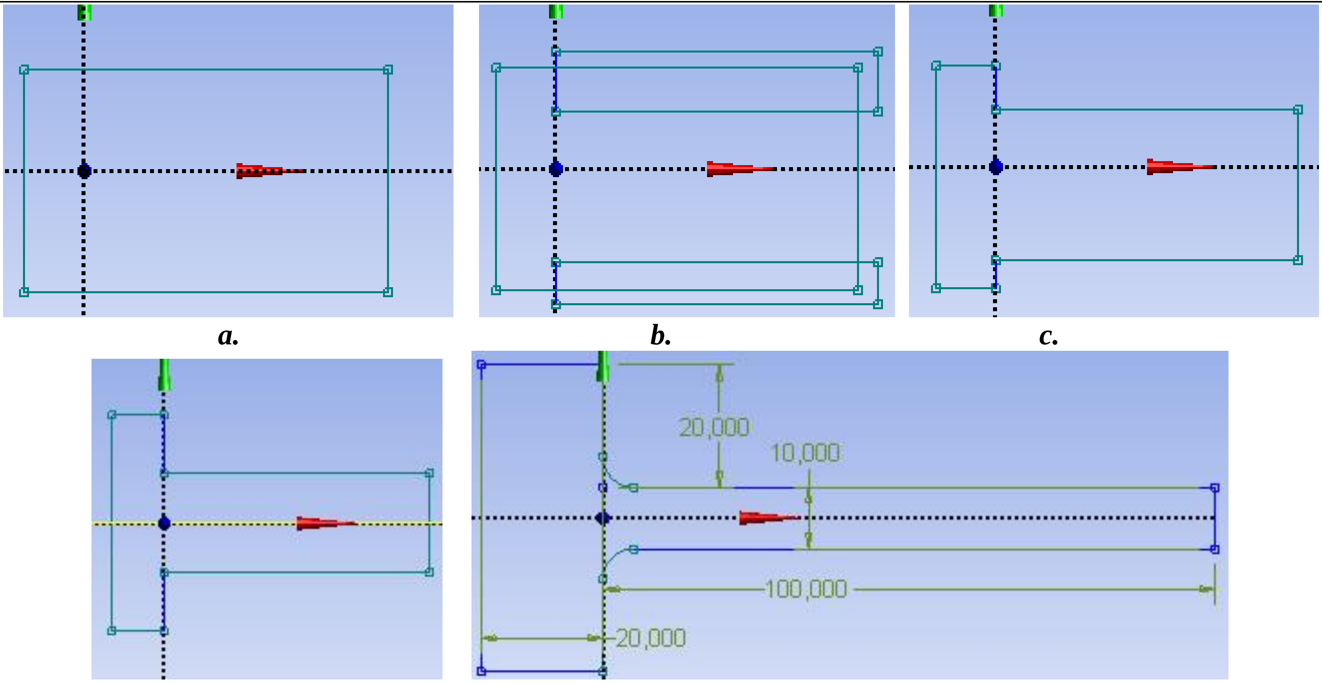
For FEA, the strength characteristics of the C55 steel are:

- longitudinal modulus of elasticity, E = 210000 N / mm^2;
- Poisson's ratio, ν = 0,3.

Average working temperature of the subassembly, T_0 = 20 ° C.

C.PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project
Creating of the project   Unsaved Project - Workbench: Toolbox: ↵⊟ Analysis Systems: ↵↵ Static Structural (the window with project modules appears automatically); [change name, Static Structural].   *Setting of problem type (2D)*   ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options, ↵Analysis Type, [selecting from drop down list ↵▼, ↵2D] → [close the window ↵X].   *Saving of the project*   ↵ Save As… → Save As, File name: [enter name, FEA-A.3] → ↵ Save.
C.2 Modelling of material and environment characteristics
→ Project Schematic → ↳ Engineering Data ✓ → ↵ Edit… → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [selecting from drop down list C (Unit) cu / with ↵▼, ↵▼], [enter in column B (Unit) valoarea, 210000] → ↵ Update Project → ↵ Return to Project (others parameters are default).
C.3 Geometric modelling
C.3.1 Model loading, DesignModeler (DM)
→ Project Schematic → ↳ Geometry → ↵ New Geometry… → ANSYS Workbench: ↵ Millimeter, ↵ OK.
C.3.2 Sketch generation
Viewing default plane (XY)   → ↵ Sketching → (Look At Face/Plane/Sketch) [automatically view of default plane, XY].   *Rectangular lines generation*   ↵Draw → ↵Rectangle → [trace rectangle line using pencil starting with, ↵ a point from left of Y axis, and finish in opposite point simultaneously with release of the mouse ↵] (fig. a) → [drawing two rectangular lines with the pencil indicator marking with, ↵ from a point of Y axis (C symbol appear), and finish in opposite point simultaneously with release of the mouse ↵] (fig. b).   *Outline beam generation*   ↵Modify → ↵ Trim → [it will be deleted by selecting with ↵ the portions of the straight segments that do not belong to the contour (fig. c)].   *Center lines in relation to the X axis*   ↵ Constraints →↵ Symmetry → [select with ↵ the X axis and then the two parallel lines with this axis to the left of the Y axis (fig. d)] → [select with ↵ the X axis and then the two parallel lines with this axis to the right of the Y axis (fig. d)].   *Dimensions*   ↵Dimensions → ↵ Semi-Automatic → [dimensions are automatically activated with ↵] → Details View, ⊟ Dimensions: 4: → [they are inserted into the boxes ☐ L1, ☐ L2, ☐ L3, ☐ L4 (fig. e)]. ↵ Display (viewing dimensions), Name: ↵☐ (it is disabled), Value: ↵☑ (is activated). ↵ Move (moving dimensions), [the dimension activates with ↵ and moves keeping the activation to the desired position] (fig. e).   *Fillet generation*   ↵Modify → ↵ Fillet → [input Radius:, radius value, 5] → [select with ↵ the connecting lines (fig. e)]

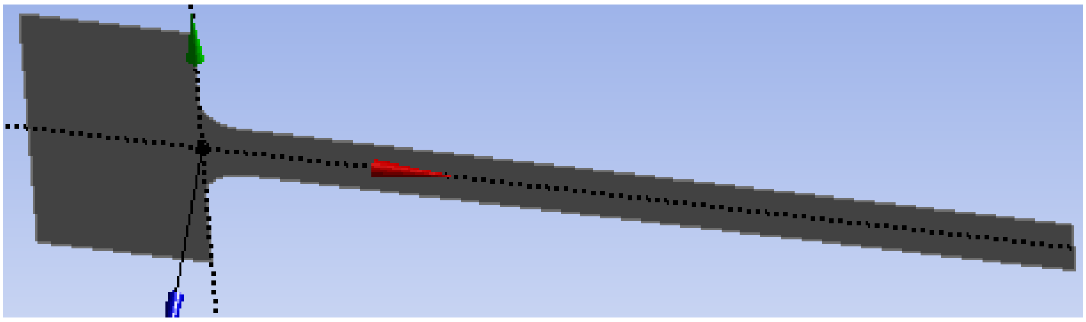


a. b. c.

d. e.

C.3.3 Surface generation

DM → ↵ Concept → ↵ Surfaces From Sketches → Details View, ⊟ **Details of SurfaceSk1**: Base Objects → ↵ Tree Outline → ⊞ XYPlane, ↵ Sketch1 → ↵ Apply; Thickness (>=0), [input thickness, 10] → ↵ Generate. ↳ Sketch1 → ↵ Hide Sketch (hide sketch) ↵ ISO (axonometric visualization).
↵ ⊞ 1 Part, 1 Body → ↵ Surface Body → Details View, ⊟ **Details of Surface Body**: Body, [input name, *Suprafață bară*].



C.3.4 Saving of geometric model

DM → ↵ (Save Project) → ↵ (Close Design Modeler).

C.4. Finite element modelling

C4.1 Launching the finite element modelling module and setting the problem type, material characteristics, and unit system

Launching the modelling module with finite elements

WB → Project Schematic → ↳ Model → ↵ M **Edit...** → [launching the *Mechanical [ANSYS Multiphysics*].

Setting the type of the problem

M → Outline → ↵ ⊞ Geometry → Details of "Geometry", ⊟ **Definition**: 2D Behavior, [select from list↵, ↵ Plane Stress (default settings)].

Setting the material characteristics

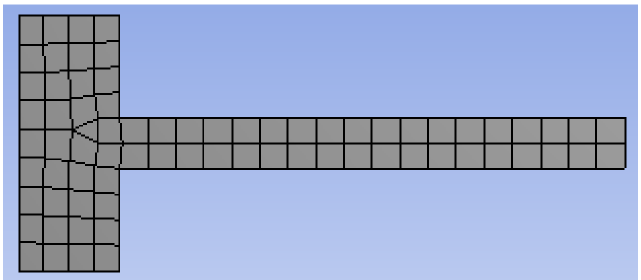
Outline → ↵ ⊞ Geometry → ↵ Suprafață bară → Details of "Suprafață bară": **Material**: Assignment, [is selected from the list ↵, ↵ Structural Steel] (usually, when there is only one material, this setting is default).

Setting the units

M → ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA).

C.4.2 Model meshing

Case I (meshing with large first order finite elements)



<u>Adopting the first order finite element</u> (with the straight line, without intermediate node)

, Outline: → ↵ Mesh → Details of "Mesh", ⊞ Advanced : Element Midside Nodes , [select from list ↵, ↵Dropped].

<u>Automaticaly meshing</u>

↳ Mesh → ↵ Generate Mesh .

<u>Visualisation of meshing statistics</u>

↵ Mesh → Details of "Mesh", ⊞ Statistics : ☐ Nodes , 106 ; ☐ Elements , 74 .

Obs. It will be continued starting with step C.4.3 and after post-processing it will be returned and re-meshing according to the following case.

Case II (meshing with small first order finite elements]n singularities areas)

<u>Adopting the first order finite element</u> (with the second order line, with intermediate node)

↵ Mesh → Details of "Mesh", ⊞ Advanced : Element Midside Nodes , [selecting from the list ↵, ↵ Kept].

<u>Setting global meshing</u>

↵ Mesh → Details of "Mesh", ⊟ Defaults : ☐ Relevance , [modifying with ↵ valoarea / value, 100].

<u>Setting local meshing to a point</u>

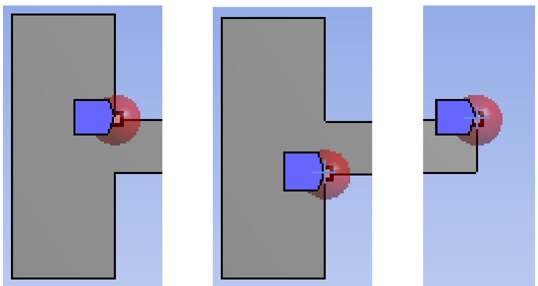
↳ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, ⊟ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ upper corner(fig. a)] → ↵ Apply; Sphere Radius, ↵Please Define → [input value 5]; Element Size, ↵Please Define → [input value, 1].

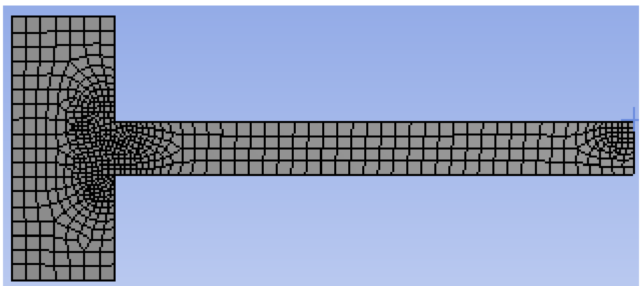
↳ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, ⊟ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ lower corner (fig. b)] → ↵ Apply; Sphere Radius, ↵Please Define → [input value, 5]; Element Size, ↵Please Define → [input value, 1].

↳ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, ⊟ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ point of application of the force (fig. c)] → ↵ Apply; Sphere Radius, ↵ Please Define → [input value, 5]; Element Size, ↵Please Define → [input value, 1].

<u>Reviewing of meshing statistics</u>

: Update → Outline: ↵ Mesh → Details of "Mesh", ⊞ Statistics : ☐ Nodes , 1954; ☐ Elements , 601 .



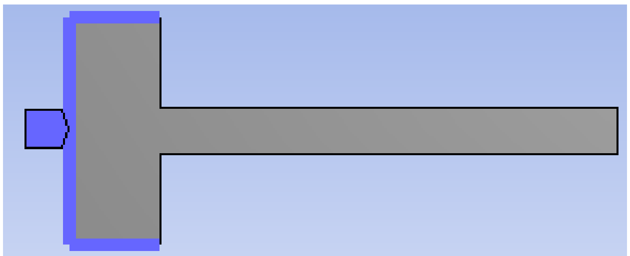


a. *b.* *c.* *d.*

Obs. It will be continued from step C.4.5

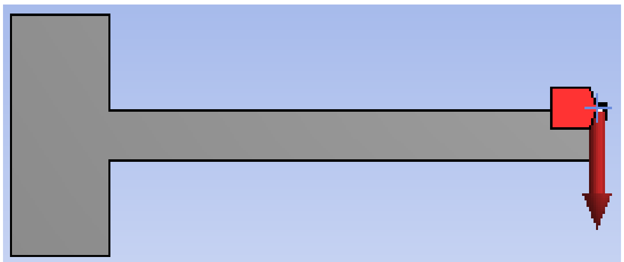
C.4.3 Supports and restraints modelling (fixed support)

, Outline: ↲ ⊟···? **Static Structural (A5)** → ↲ Supports → ↲ Fixed Support → Details of "Fixed Support", Geometry → ↲ (activating line selection filter) → [selecting with Ctrl+↲ fixed edges] → ↲ Apply.



C.4.4 Loads modelling

, Outline: ↲ ⊟···? **Static Structural (A5)** → ↲ Loads → ↲ Force → Details of "Force", ⊟ **Scope**: Geometry → ↲ (activating point selection filter) → [selecting with ↲ peak] → ↲ Apply; ⊟ **Definition**: Define By, [select from list ↲, ↲Components], ↲ Y Component, ↲0, N (ramped) → [input value, -1000].



C.4.5 Saving the project

: ↲ File → Save Project....

D.SOLVING THE FEA MODEL

D.1 Setting the results

Selecting the total displacements

, Outline: ↲ ⊞···? **Solution (A6)** → ↲ Deformation → ↲ Total.

Selecting the normal stress on X axis

↲ ⊟···⚡ **Solution (A6)** → ↲ Stress → ↲ Normal → Details of "Normal Stress", ⊟ **Definition**: Orientation, [select from list ↲, ↲X Axis] (default).

Selecting the tangential stress

↲ ⊞···? **Solution (A6)** → ↲ Stress → ↲ Shear.

Selecting the equivalent stress

↲ ⊞···? **Solution (A6)** → ↲ Stress → ↲ Equivalent (von-Mises).

Selecting the structural error

↲ ⊞···? **Solution (A6)** → ↲ Stress → ↲ Error.

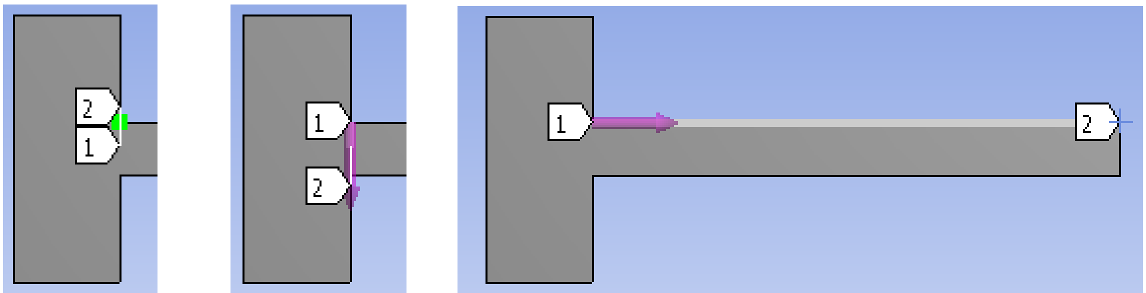
Selecting the normal stress on upper edge

Line generation: **Model (A4)** → ↲ Construction Geometry → ↲ Path → Details of "Path", ⊟ **Start**: ↲ Location, → ↲ (activating point selection filter) → [selecting with ↲ upper corner(fig. a)] → ↲ Apply (fig. b); ⊟ **End**, ↲Location, ↲Click to Change → ↲ (activating point selection filter) → [select upper right peak (fig. c)] → ↲ Apply.

Setting the normal stress on generated line

↵ Solution (A6) → ↵ Stress → ↵ Normal → ↵ Normal Stress 2 → Details of "Normal Stress 2", ⊟ Scope: ↵ Scoping Method, [selecting from list with ↵, ↵ Path]; ↵ Path, [selecting from list ↵, ↵ Path].

Setting the equivalent stress on generated line: ↵ ⊟ Solution (A6) → ↵ Stress → ↵ Equivalent (von-Mises) → Details of "Equivalent Stress 2" → ↵ ⊞ Scope: ↵ Scoping Method, [selecting from list ↵ , ↵ Path]; Path, [selecting from list with ↵, ↵ Path].



a. ***b.*** ***c.***

D.2 Launching the solving module

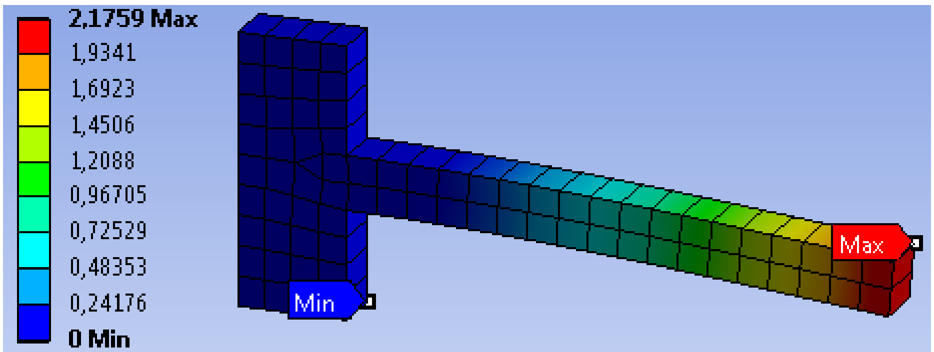
: ↵ ⊞ Solution (A6) → ↵ Solve .

E. POST-PROCESSING OF RESULTS

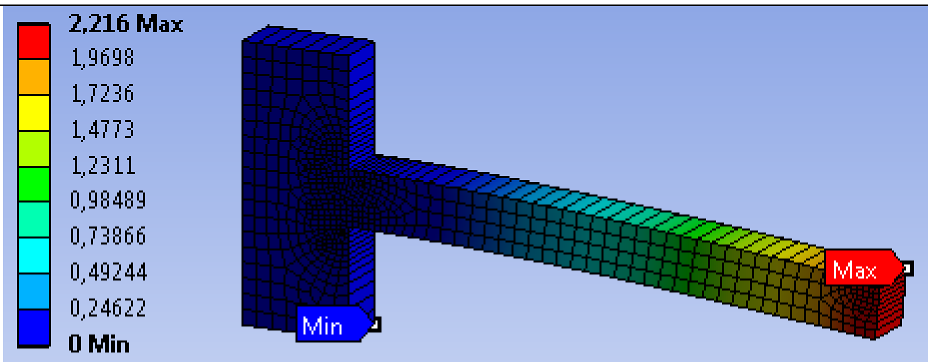
E.1. Viewing the total displacement field

, Outline: → ↵ ⊞ Solution (A6) → ↵ Total Deformation; ↵ ISO (axonometric visualization); ↵ → [selecting from the list ↵, ↵ Smooth Contours] (visualization of smooth contours) ; ↵ → [selecting from the list ↵, ↵ Show Elements] (visualization of finite elements) ; ↵ Result → [selecting from the list ↵, ↵ 3, (Auto Scale)]; ↵ MAX (marking the node with the maximum total displacement); ↵ MIN (marking the node with the minimum total displacement).

Graph → ↵ Animation (visualisation af animation).



Case I



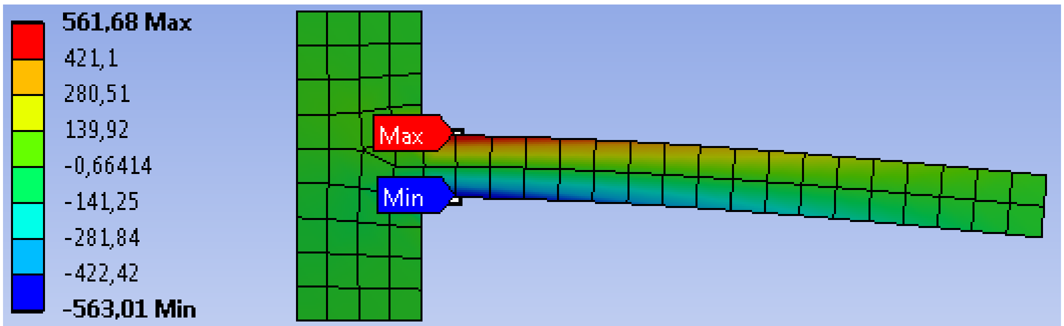
Case II

E.2. *Viewing stress field*

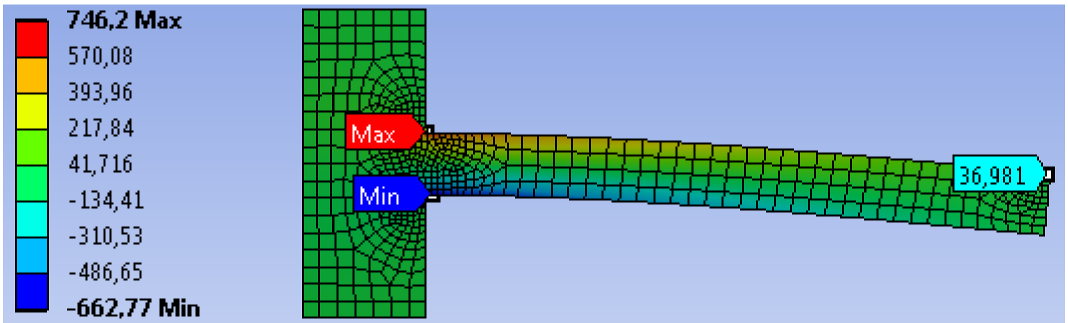
E.2.1 Viewing the displacement field normal at the X axis

Viewing the global field

, Outline: → ↵ ⊞ ? Solution (A6) → ↵ ✓ Normal Stress; [selecting with ↳ grafical model] → ↵ View → Front.



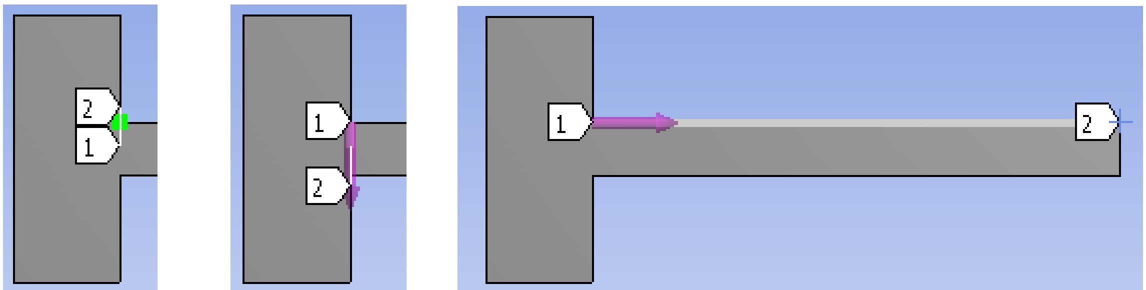
Case I



Case II

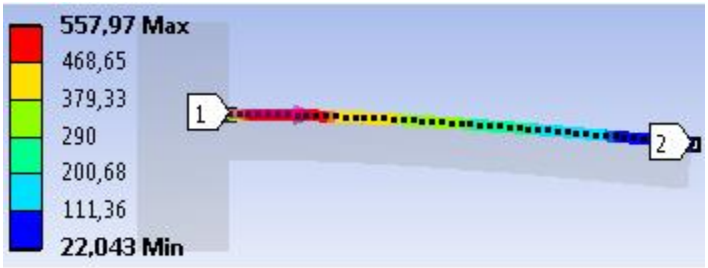
Viewing the field on a line

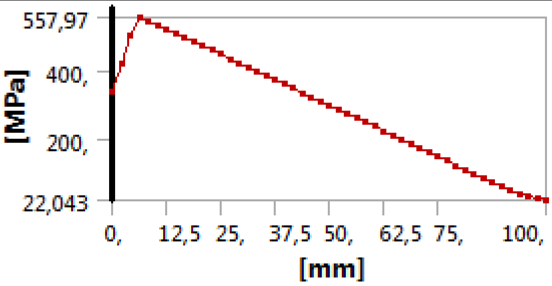
↳ Normal Stress 2 → ↵ Evaluate All Results (fig. d,e).



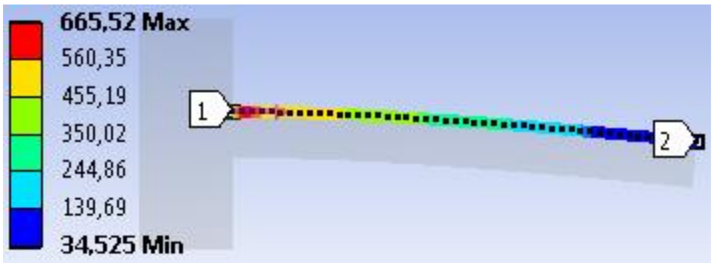
a. ***b.*** ***c.***

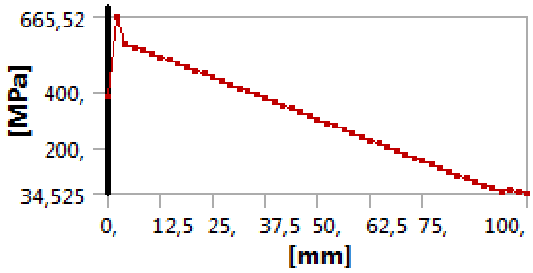
Case I





Case II

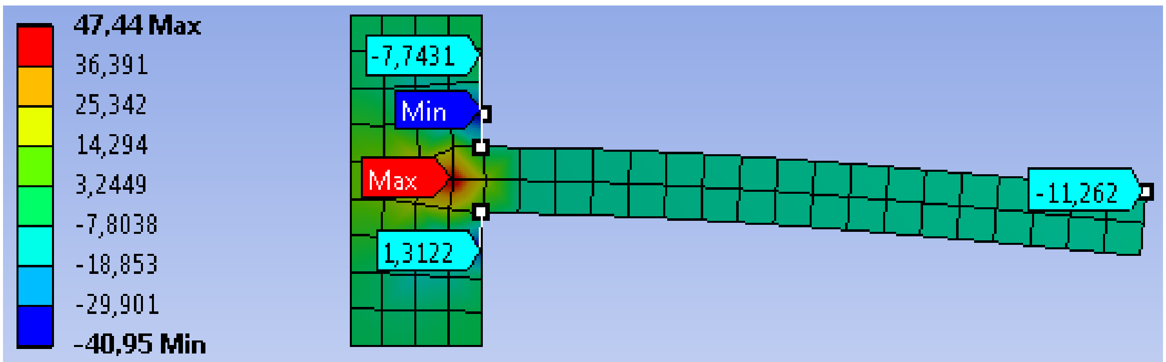




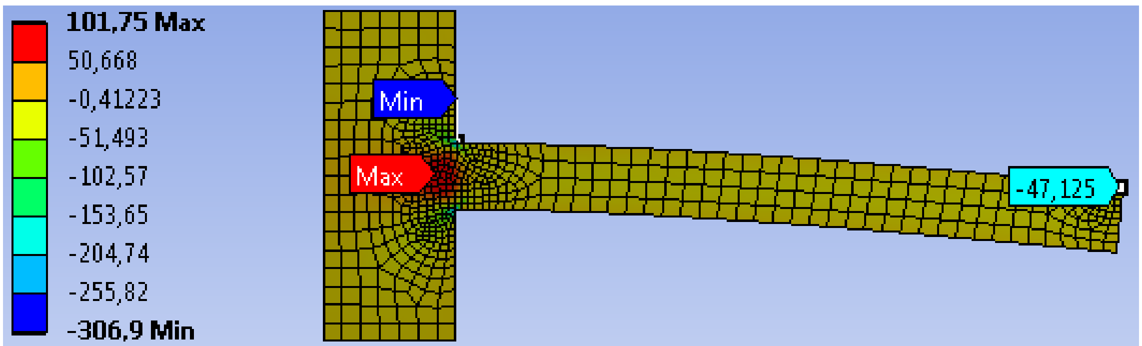
d. *e.*

E.2.2 Viewing the tangential stress field

Outline: → Solution (A6) → Shear Stress.



Case I

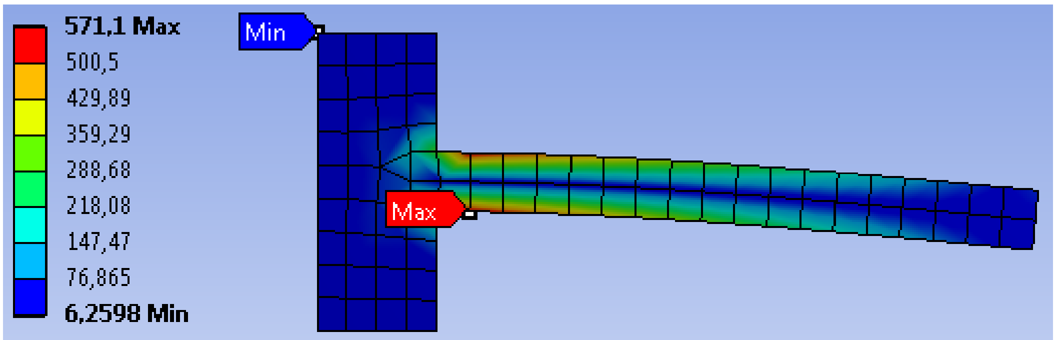


Case II

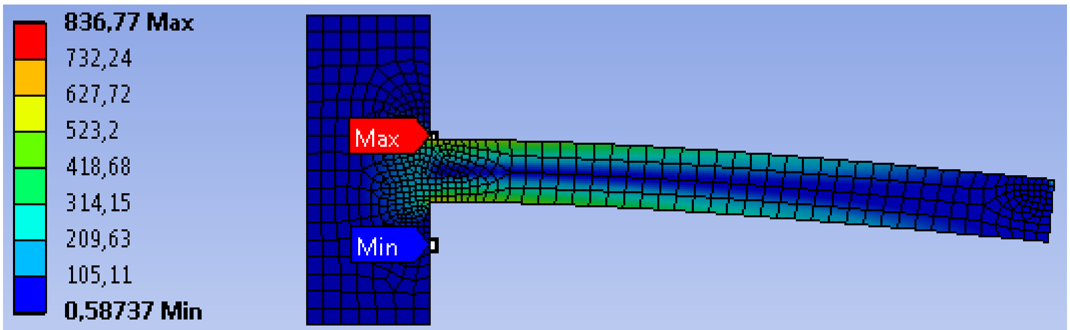
E.2.3 View the equivalent stress field (von Mises)

, Outline: → ↵ ⊞ ? Solution (A6) → ↵ Equivalent Stress.

Case I



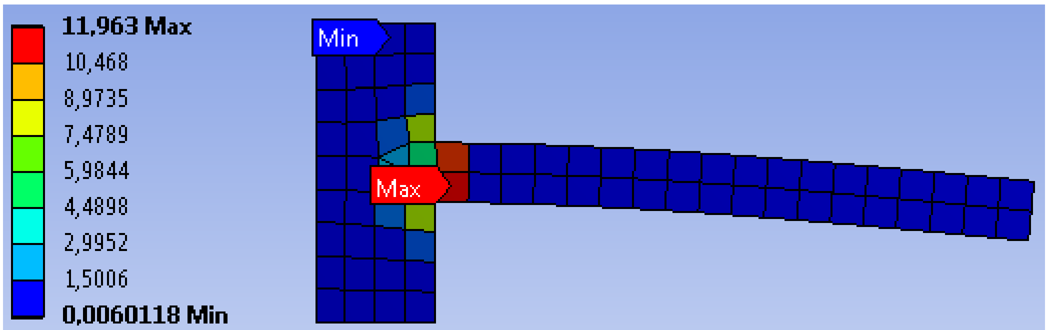
Case II



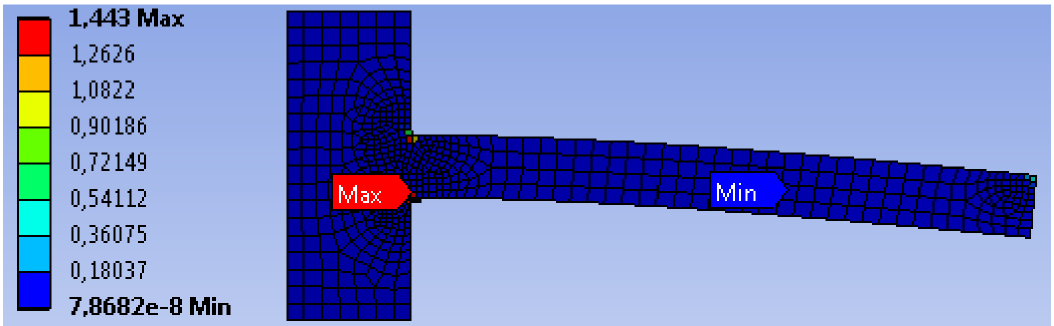
E.2.4 Viewing structural error fields

, Outline: → ↵ ⊞ ? Solution (A6) → ↵ Structural Error.

Case I



Case II



F. RESULTS ANALYSIS

F.1 The theoretical (analytical) calculation model

The classical analytical studies on the analysis structure (cantilever beam) are synthesized in the calculation of the following parameters (see application of FEA-A.1 subchapter F.1): maximum displacement, δ = 2 mm, maximum normal bending tension (according to Navier's relation) , $\sigma_{î}$ = 600 MPa, the maximum shear tangential stress (according to Juravschi's relation), τ_f = 15 MPa.

F.2 Comparison and evaluation of results

Following the analysis of the results obtained as a result of the modeling and FEA (subchapter E) and the use of the classical computational relations (subcap. E and F.1) obtained under the conditions of the materials strength hypotheses, the following are highlighted:

- The maximum total displacement, 2,179 mm (case I) or 2,216 mm (case II), obtained with FEA (E.1), is almost equal to the displacement (2 mm) obtained from the theoretical analytical model (subchapter F.1).
- The maximum normal stress in the X direction, -563.2 MPa (case I) or 746.2 MPa, obtained by finite element analysis (subchapter E.2.2) has a deviation of -6% (case I) or 24.36 % MPa (case II) against the maximum normal stress (600 MPa) theoretical (subchapter F.1).
- The shear stress distribution (E.2.3) shows maximum values, 47.44 MPa (case I) or 101.75 MPa (case II), in the recessed area is 3.12 times (case I) or 6.78 or (case II) against the theoretical value, 15 MPa.
- The equivalent stress (von Mises) has the maximum value, 571.1 MPa (case I) or 836.72 MPa (case II) in the compressed and stretched area, respectively; it is observed that with the increase of the meshing fineness (case II) the value of the equivalent stress (von Mises) deviates by 39.4% due to the corner singularity (connection with null radius).

F.3 Accuracy analysis based on structural error

In subchapter. E.2.3 the structural error with the maximum value of 11.96 mJ (case I) or 1,443 mJ (case II) is highlighted; the maximum value in case I shows maximum errors of the stress in the fixing area.

The structural error is determined as the difference of the deformation energies calculated using the average stresses associated with the finite element and the nodal stresses. The fineness of increased discretization leads to reduced structural error values and, therefore, it can be used on the one hand, as a global indicator of the discretization fineness, in the rediscretion of the entire structure and, on the other, as a local indicator of meshing fineness at local rediscretion. In order to assess the accuracy of the stress type results, the field of structural error is analyzed, following a uniform distribution with reduced (preferably subunit) values of the structural error for acceptable accuracies; the areas where the structural error is increased in order to increase the precision of the results (the decrease of the structural error) will be made local rediscretion (subchapter E.2.3).

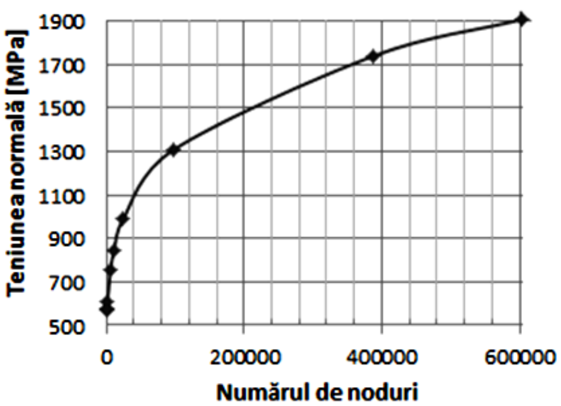
F.4 Analysis of convergence on X axis

In order to highlight the effects of corner singularities (zero radius fillet) and concentrated force (point action), the model will be analyzed with various meshings, following the values of normal stress in the X direction, especially in the areas with singularities. For this purpose, the succession of modifying the fineness of meshing at the global level will be followed (the second order finite element set above will be kept):

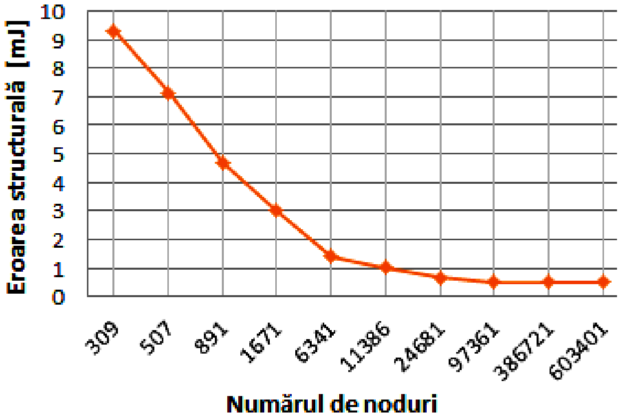
↵ Mesh → Details of "Mesh", ⊟ Sizing: Use Advanced Size Function, [se selectează din listă cu / selecting from the list with ↵▾, ↵ Off];

↲ Element Size , [se introduce valoarea dimensiunii elementului finit conform coloanei întâi din tabelul de mai jos / input the finite element size value according to the first column in the table below]. ↲ Mesh → ↲ Generate Mesh . ↲ Mesh → Details of "Mesh", ⊞ Statistics : [se evidenţiază numărul de noduri din caseta / showing nodes number, Nodes (coloana a treia / third column) şi numărul de elemente number of elements, Elements (coloana a doua / second column)].

Dimension EF [mm]	Number of EF	Number of nodes	/Normal stress [MPa]	Structural error [mJ]
5	80	309	572,96	9,3176
4	140	50	570,96	7,1435
3	258	891	577,32	4,712
2	500	1671	605,4	2,99
1	2000	6341	757,07	1,389
0,75	3200	11386	847,47	1,0124
0,5	8000	24681	990,35	0,651
0,25	32000	97361	1306,7	0,5
0,125	128000	386721	1737,1	0,5
0,1	200000	603401	1906,5	0,5

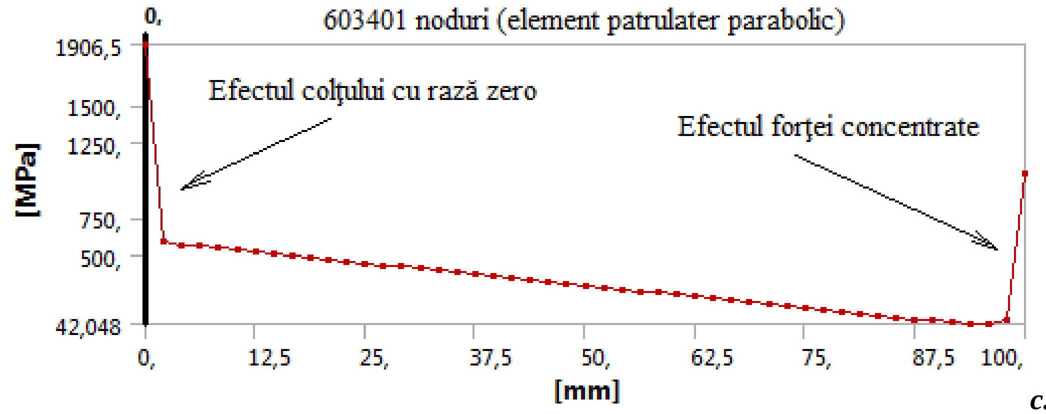


a.



b.

At the corner point with singularity of the normal stress, its values increase with the increase of the number of nodes (there is no asymptote to tend to). The structural error decreases with the increase of the discretization fineness but at higher values of the nodes it has low values and it remains quasi-constant and the values of the normal stress increase non-asymptotically, which demonstrates the inconsistency of the process in the corner area with singularity (fig. A, c). The same situation specific to the singularity of the stress is observed in the area of application of the concentrated force (fig. C)



c.

G. CONCLUSIONS

The modeling and analysis with finite elements of this paper were done more with didactic purpose aiming, on the one hand, the initiation of the user with the main stages of developing an application of FEA in ANSYS Workbench and, on the other hand, the comparison. and evaluating the results obtained from FEA with different shapes and sizes of finite elements.

The adopted FEA model leads to coarse deviations from the exact solution for the linear triangular finite element as opposed to the quadratic finite element model which shows a convergence with very small deviations.

The analysis of the results, in particular, of the stresss, for discretizations with increased fineness, shows that in the area with t singularities, although the structural error decreases to allowable values which would show a good accuracy, the values of the stresss do not converge towards the cvasiexact value, but they grow non-asymptotically.

The FEA model studied in this paper is inefficient in terms of modeling possibilities offered by the ANSYS platform because the connection area in the recess is null radius (theoretical case) and the force is concentrated at one point (also theoretical case). . These aspects are avoided in the application of FEA-A.2.

Application: FEA-A.4
Bearing inner ring

KEY WORDS
Linear static analysis, Planar geometric model, Axial-symmetrical state of stresses, Linear material, 2D geometric model (plane), 2D finite element, Non-linear finite element (parabolic), Axial symmetry, Radial symmetry, Re-meshing, Machine element, Bearing ring

CONTENT

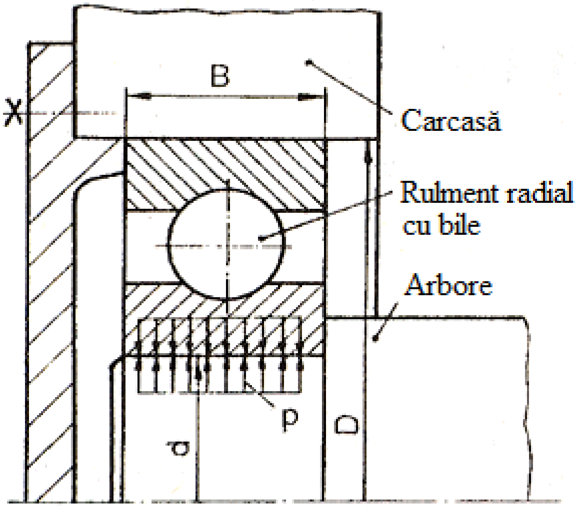
A. PROBLEM DESCRIPTION

A.1. Introduction

The study of the elements of mechanical systems with a common axis of symmetry for the geometric domain, material characteristics, loading and boundary conditions can be carried out using axial-symmetric models. Their structures, from a geometrical point of view, are reduced to plane geometric models associated with axial semisections which, from a physical point of view, synthesize the spatial states of stresses and deformations related to a cylindrical coordinate system with the dimension axis identical to the axis of symmetry.
The cases of practical application of the analysis with finite axial-symmetrical elements are multiple, noting with increased frequency the problems with homogeneous structures of revolution with respect to an axis, evenly distributed circumferentially distributed. Thus, the analysis of the structures of the three-dimensional elements of the machines, installations and machines, which comply with the conditions specified above, is performed by means of a plan model with a number of degrees of freedom much reduced compared to the three-dimensional model.

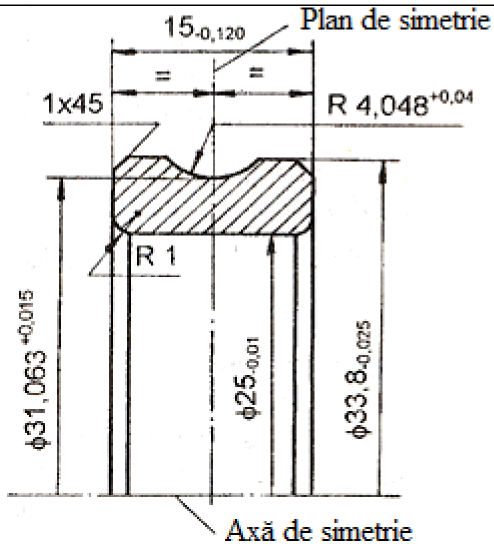
A.2. Application description

The figure below shows the radial ball bearing assembly of a shaft extension system of a speed reducer. In order to obtain the optimal functional requirements (good centering, attachment of the ring to the shaft / housing) the bearing rings are assembled pressed on the shaft head section and in the bore of the housing. As a result of the presses assemblies (with their own tightening), taking into account that the shaft and the housing have radial rigidity much larger than of the rings, radial displacements of the tread points appear in order to reduce the play in the bearing. Thus, under increased tightening conditions it can be reached after the assembly to cancel the bearing from the bearing and, therefore, to the improper operation, with high friction, which lead to overheating and shortening the life of the bearing. The analysis of the inner ring / shaft and outer ring / housing adjustments results in increased tightness in the assemblies pressed from the inside.



A.3. The application goal

In this application, using the finite element analysis, the study of the pressed assembly between the inner ring of the radial ball bearing and the shaft of a speed reducer is presented. Since, the shaft is full cross-section and, therefore, with increased radial rigidity, it is considered, for the study of said assembly, only the inner ring of the radial ball bearing (6205), executed in the precision class PO



with the normal radial game with the value in [0.01; 0.02] mm. The inner ring of this bearing with the shape and dimensions shown in the attached figure is made of bearing steel, the mark RUL1, with the modulus of longitudinal elasticity E = 2.1 10^5MPa, the coefficient of transverse contraction $\upsilon = 0.3$ and the density, $\rho = 7800$ kg / m^3.

In this study, it is intended, for the concrete case described above, the determination of data on displacements and stresses in the inner ring, the change of the tread pattern, the pressure on the mounting surface and the mounting / dismounting force of the ring on the shaft. These can also be obtained taking into account the fact that the inner ring pressed on the shaft is rotating with the speed n = 4000 rot / min.

B. THE FEA MODEL

B.1. The model definition

Since the geometric and loading structure is symmetrical with respect to an axis as well as with a transverse plane, a plane (2D) model determined by the section of the radial semisection through the inner ring is adopted for analysis.

Thus, without losing accuracy, the problem to be solved falls within the axial symmetric state of tensions and the simplest possible model is adopted, which implies:

• the flat geometric shape,
• discretization with 2D nonlinear finite elements (parabolic),
• linear behavior of the material,
• adoption of constraints associated with symmetry properties,
• external load by forced displacement.

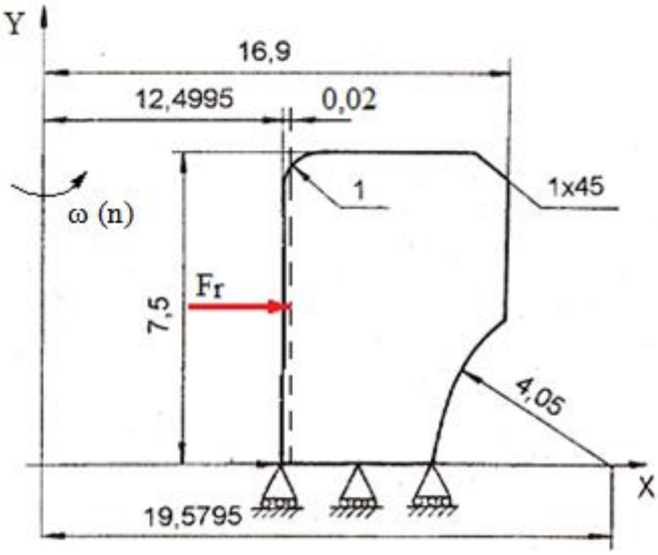
B.2. The analysis model description

The figure below shows the FEA model associated with the plane geometric model of the axial semisection considered in the XY plane with the Y axis (symmetry axis) of the structure to be analyzed. In addition, it is observed that the considered plane domain and its deformed state are symmetrical with respect to the plane XZ perpendicular to the axis of rotation (Y) and is identical to the plane of symmetry of the tread.

The imposed (boundary) boundary conditions, in accordance with the considered symmetries, involve free radial displacements of the model points on the X axis and cancel the displacements along the Y axis.

The load of the structure is realized by the imposed displacement of the inner edge with the value of the maximum radial tightening, 0.02 mm, calculated for the adjustment H7$\binom{+0{,}021}{0}$ / r6 $\binom{+0{,}041}{+0{,}026}$; consequently, the force Fr appears to be determined

In addition, the structure to be considered is considered to rotate around the Y axis with the angular velocity $\omega = \pi n / 30 = 418.88$ rad / s.



B.3. Characteristics of the material and the environment

For linear static analysis the following resistance characteristics of RUL1 material are considered:

- longitudinal modulus of elasticity, E = 210000 N / mm^2;
- Poisson's ratio, ν = 0,3.

Average working temperature of the subassembly, T_0 = 20 ° C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project

Unsaved Project - Workbench: Toolbox: ↵ ⊟ Analysis Systems: ↵↵ Static Structural (the window with project modules appears automatically); [change name, Static Structural].

Setting of problem type (2D)

↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options, ↵ Analysis Type, [selecting from drop down list ↵▼, ↵2D] → [close the window ↵ ✕].

Saving of the project

↵ Save As... → WB Save As, File name: [enter name, FEA-A.4] → ↵ Save.

C.2 Modelling of material and environment characteristics

WB → Project Schematic → ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [selecting from drop down list C (Unit)with ↵▼], [enter in column B (Unit) value, 210000] → ↵ Update Project → ↵ Return to Project (others parameters are default).

C.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

WB, Project Schematic: ↳ Geometry → ↵ DM New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.

C.3.2 Sketch generation

Viewing default plane (XY)

DM, Tree Outline: ↵ Sketching → (Look at face/Plane/Schetch) [automatically view of default plane, XY] ;

Rectangular lines generation

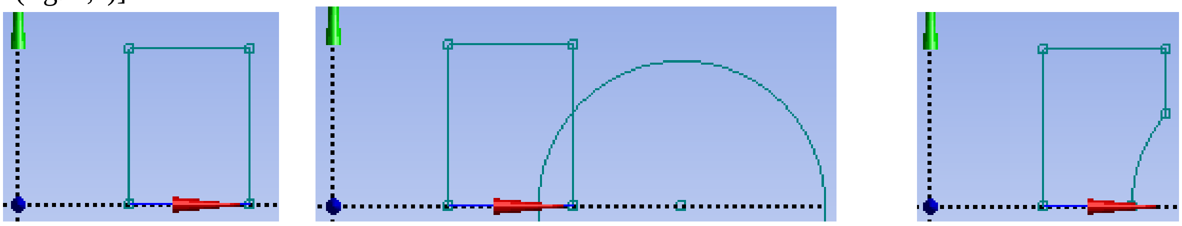
↵Draw → ↵ Rectangle → [trace rectangle line using pencil starting with, ↵ a point from left of Y axis, and finish in opposite point simultaneously with release of the mouse ↵] (fig. a)].

Generating a circle arc

↵ Circle → [the circular line is generated by ↵ marking the center on the x-axis (coincidence symbol C appears), moving in the radial direction and releasing ↵ on the contour, fig. b].

Triming lines at the edge

↵Modify → ↵ Trim → [activate - desactivate with ↵ the option Ignore Axis (☑/☐]) → [marking with ↵ the part to be cut (fig. b,c)]



a. b. c.

Dimensioning

Sketching Toolboxes: ↵ Dimensions → ↵ Horizontal → [selecting two entities (points, lines, axes) with ↵ and the dimensions appear automatically (fig. a)] → Details View, Dimensions: 1 : ☐ H → [introducing the dimension (fig. a)].

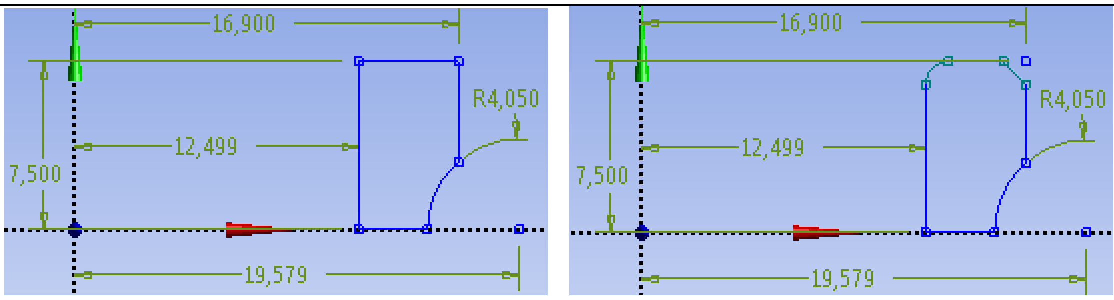
↵ Vertical → [selecting with ↵ two entities (points, lines, axes) with ↵ and the dimensions appear automatically (fig. a)] → Details View, Dimensions: 1 : ☐ V → [introducing the dimension (fig.a)]. ↵ Radius → [selecting with the circle arc and the dimensions appear automatically (fig. a)] → Details View, Dimensions: 1 : ☐ R → [input the radius value (fig.a)].

Fillet generation

↵Modify → ↵ Fillet → [input Radius: radius value, 1] → [select with ↵ the connecting lines (fig.b)].

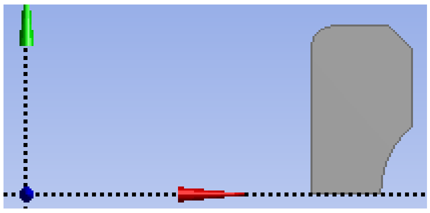
Chamfer generation

↵Modify → ↵ Chamfer → [input in Length: lenght value, 1] → [select with ↵ the connecting lines (fig.b)].



a. *b.*

C.3.3 Surface generation



: ↵ Concept → ↵ Surfaces From Sketches → Details View, ⊟ Details of SurfaceSk1: Base Objects → Tree Outline : ⊞ XYPlane, ↵ Sketch1 → ↵ Apply. ↵ Generate.

Tree Outline: ↵ ⊞ 1 Part, 1 Body → ↵ Surface Body → Details View, ⊟ Details of Surface Body: Body, [input name, *Suprafață radială*].

C.3.4 Save of geometric model

: ↵ (Save Project) → ↵ (Close).

C.4. Finite element modelling

C.4.1 Launching the finite element modelling module and setting the problem type, material characteristics, and unit system

Launching the modelling module with finite elements

→ Project Schematic → ↳ Model → ↵ Edit... → [launching the *Mechanical [ANSYS Multiphysics*].

Setting the type of the problem

→ Outline → ↵ ⊞ Geometry → Details of "Geometry", ⊟ Definition: 2D Behavior, [select from list ↵, ↵ Plane Stress (default settings)].

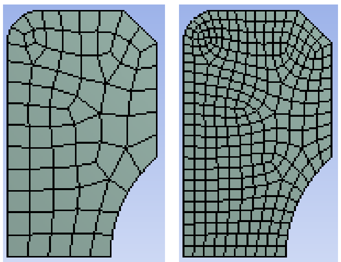
Setting the material characteristics

Outline → ↵ ⊞ Geometry → ↵ Suprafață bară → Details of "Suprafață bară": Material : Assignment, [is selected from the list ↵, ↵ Structural Steel] (usually, when there is only one material, this setting is default).

Setting the units

: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA).

C.4.2 Model meshing



a. *b.*

Automaticaly meshing (with default parameters)

, Outline: ↳ Mesh → Generate Mesh (fig. a).

Setting re-meshing

↳ Mesh → ↵ Insert → ↵ Refinement → Details of "Refinement" - Refinement, ↵ ⊟ Scope : ↵ Geometry → ↵ (setting the surface selection filter) → [select with ↵ the surface]; ⊟ Definition : ↵ Refinement, [indexing with ↵ the number of successive re-meshings, 1 (fig. b)].

Automaticaly re-meshing

↳ Mesh → Generate Mesh.

Visualisation of meshing statistics

↵ Mesh → Details of "Mesh", ⊞ Statistics: Nodes, 921; Elements, 284.

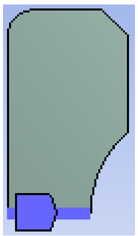
C.4.3 Supports and restraints modelling

Generating the fixed support

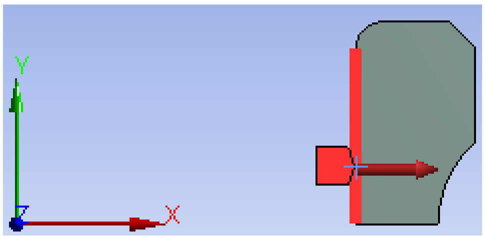
, Outline: ↵ Static Structural (A5) → ↵ Supports → ↵ Frictionless Support; → ↵ (activating line selection filter) → [selecting with ↵ the line from OX axes (fig. a)];

Generating the displacement support

↵ Static Structural (A5) → ↵ Supports → ↵ Displacement → Details of "Displacement", Scope: ↵ Geometry → ↵ (activating line selection filter) → [selecting with ↵ the edge (fig. b)] → ↵ Apply → Definition: ↵Define By, [selecting frommlist with ↵, ↵Components], ↵ X Component, ↵ Free, [selecting from the list ↵, Constant], [input the displacement value, 0,02] (fig. b).



a.



b.

C.4.4 Loads modelling

, Outline: ↵ Static Structural (A5) → ↵ Inertial → ↵ Rotational Velocity → Details of "Rotational Velocity": ↵ Scope : ↵ Geometry → ↵ (activating face selection filter) → [selecting with ↵ suprafaţa/ the surface]; Definition : ↵Define By, [selecting from the list ↵, ↵Components]; ↵ Y Component, [input the angular velocity value (rad/s), 418.18].

C.4.5 Saving the project

: ↵ File → Save Project...

D. SOLVING THE FEA MODEL

D.1 Setting the results

Selecting the directional displacements on X axis

, Outline: ↵ Solution (A6) → ↵ Deformation → ↵ Directional → Details of "Directional Deformation":

↵ Definition: Orientation, [selecting from the list with ↵, ↵X Axis].

Selecting the equivalent stress

↵ Solution (A6) → ↵ Stress → ↵ Equivalent (von-Mises).

Selecting the normal stress on X axis

↵ Solution (A6) → ↵ Stress → ↵ Normal → Details of "Normal Stress", Definition : ↵Orientation, [selecting from the list ↵, ↵X Axis].

Selecting the normal stress on Y axis

↵ Solution (A6) → ↵ Stress → ↵ Normal → Details of "Normal Stress 2", Definition : ↵Orientation, [selecting from the list ↵, ↵Y Axis].

Selecting the normal stress on Z axis

↵ Solution (A6) → ↵ Stress → ↵ Normal → Details of "Normal Stress 3", ⊟ Definition : ↵ Orientation , [selecting from the list ↵ , ↵ Z Axis].

Setting the reaction force corresponding to the imposed displacement

↵ Solution (A6) → ↵ Probe → ↵ Force Reaction → Details of "Force Reaction", ⊟ Definition : Boundary Condition, [selecting from the list ↵ , ↵ Displacement].

D.2. Launching the solving module

: ↵ ⊞ Solution (A6) → ↵ Solve .

E. POST-PROCESSING OF RESULTS

E.1. Set the view mode in section / space

Sectional / spatial view setting

↵ → ↵ Tools → ↵ Options... → Options : ↵ Appearance , [activate – desactivate from the list ↵ , ↵ Beta Options].

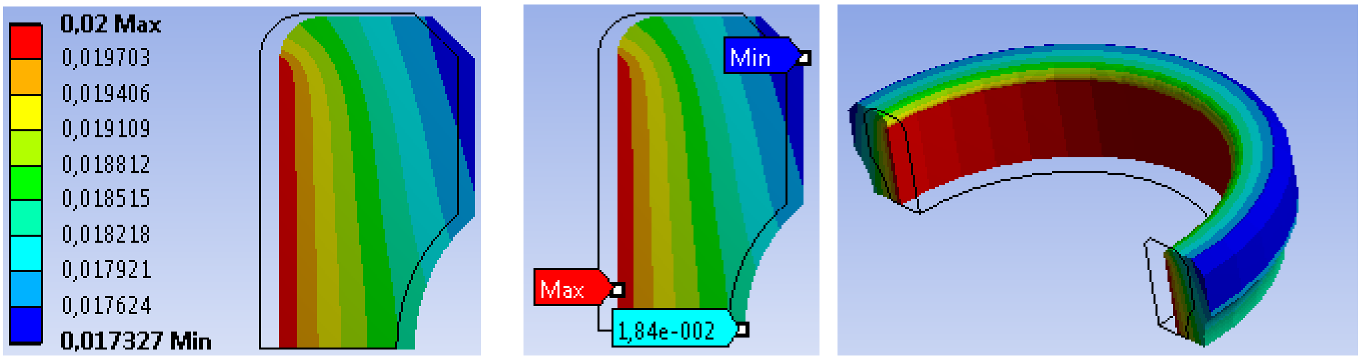
Setting the number of angular sectors

↵ → ↵ Model (A4) → ↵ Symmetry → Details of "Symmetry", ↵ ⊟ Graphical Expansion 1 (Beta) : ↵ Type , [selecting from the list ↵ , ↵ 2D AxiSymmetric]; ↵ Num Repeat [input the number of angular sectors, 19] (fig. c).

E.2. Viewing the displacement field along the X axis

, Outline : ↵ ⊞ Solution (A6) → ↵ Directional Deformation , ↵ → ↵ Show Undeformed WireFrame (undeformed model view, fig. a); ↵ Animation (animating view) .

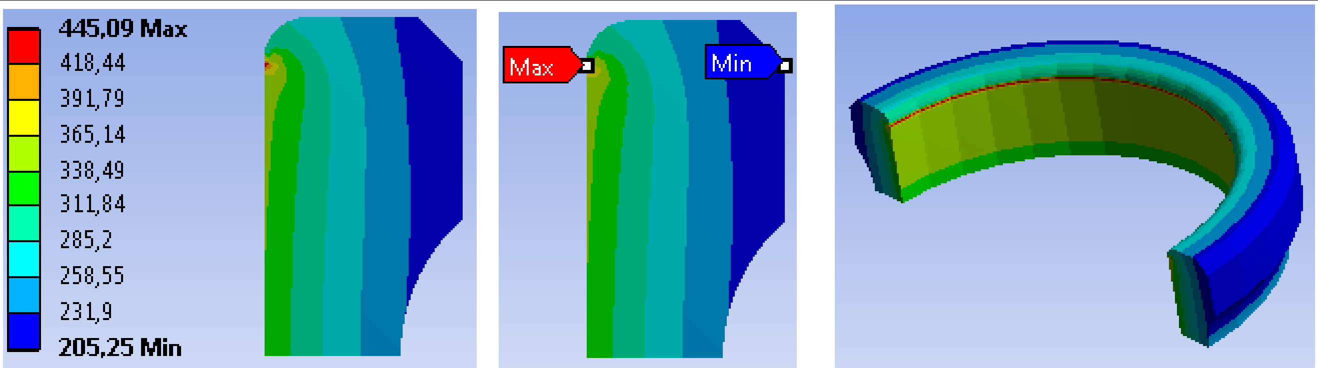
↵ Probe → [marking with ↵ the point where the value visualization is desired, 0,0184] (marking the value with label, fig. b); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. *b.* *c.*

E.3. *Viewing the equivalent stress field*

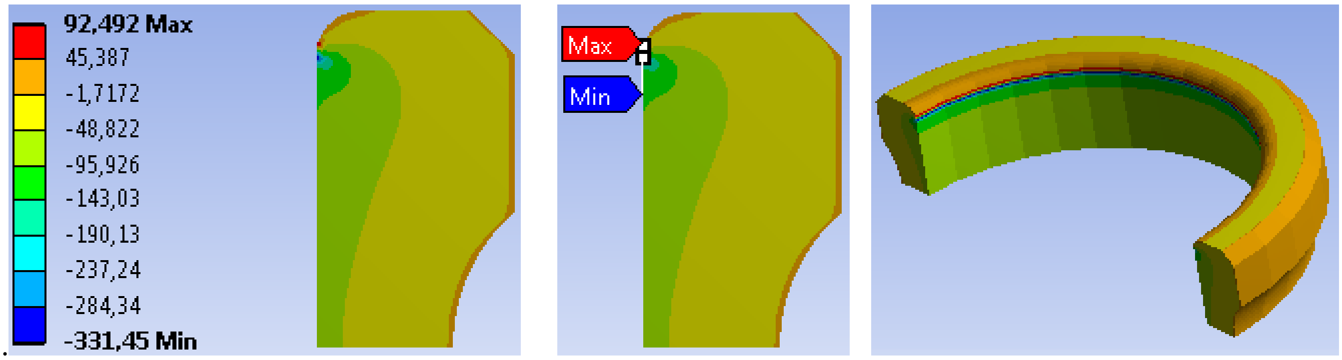
, Outline : ↵ ⊞ Solution (A6) → ↵ Equivalent Stress , ↵ → ↵ Show Undeformed WireFrame (undeformed model view, fig. a); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. *b.* *c.*

E.4. Viewing the stress field along the X axis (radial)

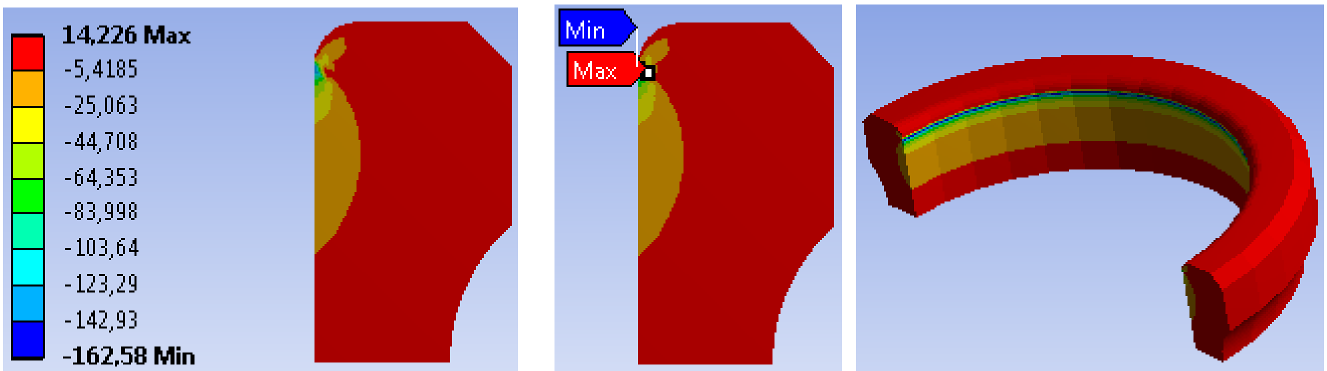
Outline: ↵ Solution (A6) → ↵ Normal Stress; ↵ → ↵ Show Undeformed WireFrame (undeformed model view, fig. a); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. *b.* *c.*

E.5. Viewing the stress field along the Y axis (axial)

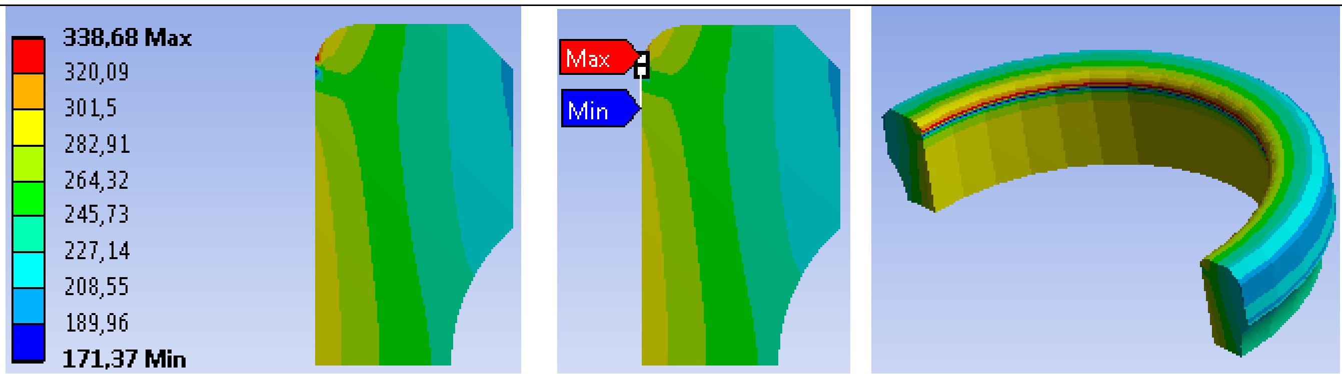
Outline: ↵ Solution (A6) → ↵ Normal Stress 2; ↵ → ↵ Show Undeformed WireFrame (undeformed model view, fig. a); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. *b.* *c.*

E.6. Viewing the stress field along the Z axis (circumferential)

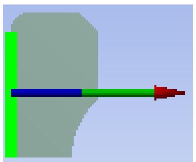
Outline: ↵ Solution (A6) → ↵ Normal Stress 2; ↵ → ↵ Show Undeformed WireFrame (view undeformed model, fig. a); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. ***b.*** ***c.***

E.7. Viewing the reaction force

Outline: ↵ Solution (A6) ⟶ ↵ Force Reaction (fig. a,b); ↵ Animation (axonometric view, fig. c).



Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]
1	1,	48197	22,627	0,

a. ***b.***

F. RESULTS ANALYSIS

Following the analysis of the obtained results (subchapter E) as a result of modeling and solving the following are highlighted:

- The radial displacement (in the direction of the X axis) in the area of adjustment of the shaft ring has the required value 0.02 mm (subchapter E.2).
- The radial displacement at the level of the tread with the value 0.0184 mm leads to the reduction of the play in the bearing (subchapter E.2, b); this displacement will be expected to be smaller than the radial bearing clearance.
- The equivalent voltage (von Mises), useful for the design of the shaft-bearing adjustment, has values increased inside with a maximum of 445.09 MPa in the starting areas of the internal connections (subchapter E.3).
- The radial tension (in the X axis direction) is compression with the maximum value -331.45 MPa, also in the starting areas of the internal connections (subchapter E.4).
- The axial tension (in the direction of the X axis) has the maximum value -162.56 MPa, also in the starting areas of the internal connections (subchapter E.5).
- The circumferential voltage (in the Z axis direction) has a maximum value of 338.68 MPa also in the starting areas of the internal connections (subchapter E.6).
- The reaction that occurs in the bore area due to the imposed radial displacement has a much larger radial component (48197 N), a very small axial component (22.627 N) and a null circumferential component.

G. CONCLUSIONS

Following the displacement fields and their maximum values, we observe the increased influence of radial displacements on the displacements of the points on the rolling path.

Conventionally, the radial stiffness of the bearing ring is defined as the ratio of the radial reaction force to the radial displacement imposed,

$$k_r = \frac{F_r}{u_r} \qquad (1)$$

which after evaluation with the values of the above model becomes k_r = 2409850 N / mm. Taking into account the linear behavior of the structure and the ratio between the radial displacement of the points in the bore area and that of the points on the rolling path, $a = u_r/u_c = 0.02 / 0.0184 = 1.09$ can be calculated according to from the minimum radial clearance the effective tightening of the shaft-bore inner ring adjustment.

The mounting / dismounting force, considering the friction coefficient $\mu = 0.2$ can be calculated with the relation,

$$F_{m/d} = \mu F_f = 0.2.\ 48197 = 9639.4 \text{ N}.$$

The pressure on the contact surface is determined by the relation $p = F / \pi D (b-2r) = 48197 / \pi\ 25\ 13 = 47.2$ MPa

Application: FEA-A.5
Diaphragm spring

KEY WORDS
Nonlinear static analysis, Thick membrane stress state, Linear material, 2D geometric model (membrane), 2D finite element, Nonlinear finite element (parabolic), Cyclic axial symmetry, Axial loading symmetry, Cylindrical coordinate system, Machine element, Diaphragm spring

CONTENT

A. PROBLEM DESCRIPTION

A.1. Introduction

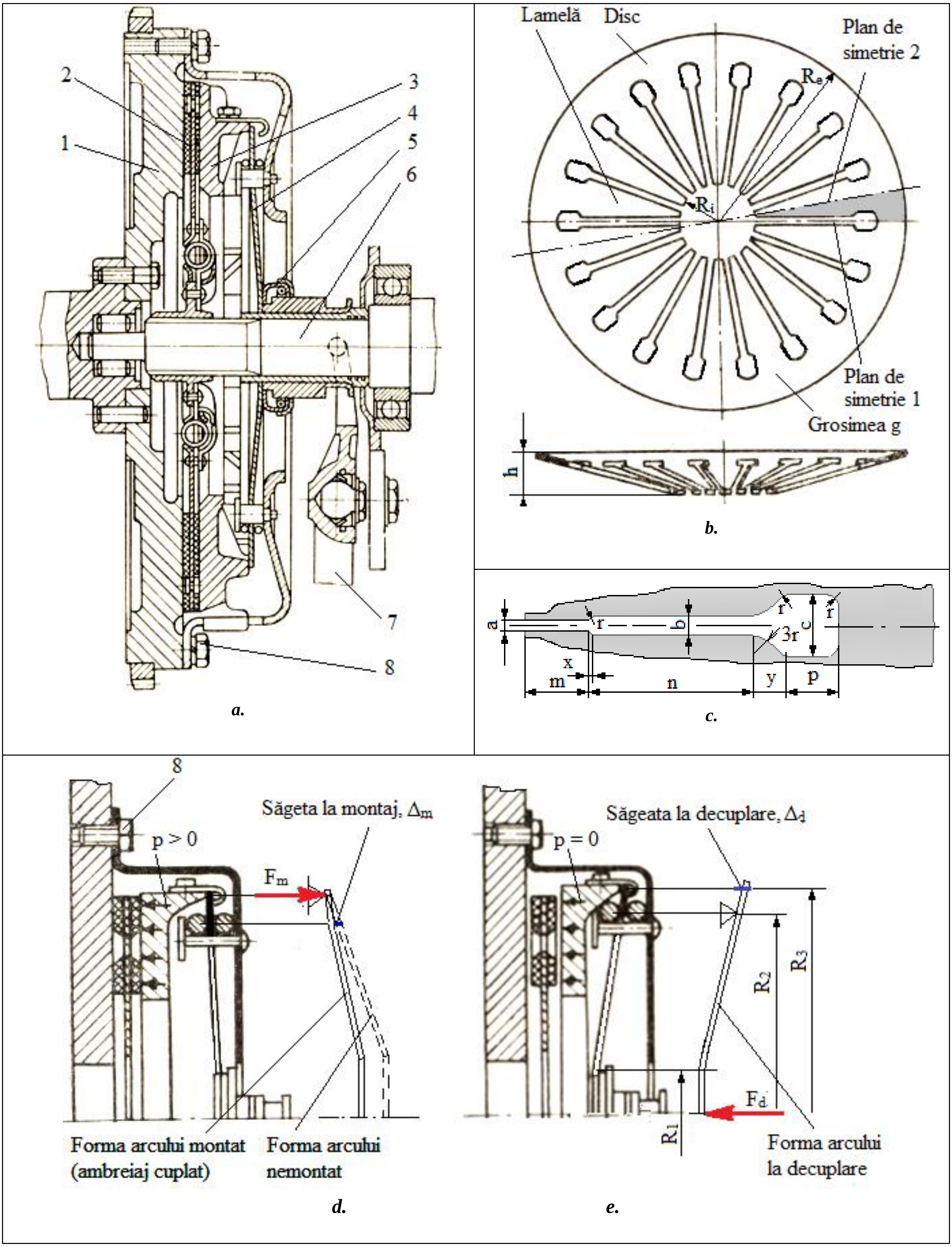
The clutch of the car is a normal intermittent mechanical coupling coupled *with the main function of decoupling-coupling* the transmission of the car in case moment of changing gears or brakes under load. In addition, secondary functions are required for optimum operation: *smooth decoupling and coupling, without shocks and vibrations; simple and easy operation, good heat transfer to the outside*; simple and technological construction; *reduced inertia of the driven parts, safe and long-lasting operation.*

The mechanical clutches based on the transmission of the torque by friction involve a controlled pressing subassembly which, in particular, in the case of small dimensions has a diaphragm spring which besides the generation of the pressing force (required for the transmission of the load) also has a functional control role .

A.2. Application description

Structure and operation of the clutch with diaphragm spring. The mechanical clutch in the figure above transmits the torque by friction from the flywheel assembly 1 and the pressure plate 3 to the disc 2 and through the groove to the main shaft of the gearbox 6. This process occurs when the lever 7 is inactive and the plate is activated. pressure 3 is pressed by the diaphragm spring 4 on the disc 2 and the steering wheel 1. At the action of the lever 7, the pressure bearing 5 presses on the diaphragm spring internally, removing the pressure disc 3 and interrupting the torque transmission. With the reduction of the pressing force on the pressure bearing the diaphragm spring returns (sometimes aided by another elastic element) and the coupling is performed.

Assembly and operation of the diaphragm spring. On the first hand, the diaphragm spring type element assures the function of generating the initial pressing force. It is mounted in the subassembly of the pressure disc 3 which is then mounted in the general assembly by means of screw assemblies 8 and, on the other part, of the displacement function of the pressure disk for decoupling. The first function involves the deformation of the outer part, similar to a disc spring with increased rigidity, by moving the area with radius R_2 with the arrow Δ_m reduced leading to an increased force F_m. The second function involves the deformation of the diaphragm spring, supported in the base area of the wings with reduced rigidity, under the action of the decoupling force F_d, it produces the decoupling by unloading the outer part of the press area, with the reduced value Δ_d.


2
1
3
4
5
6
7
8
a.
Lamelă
Disc
Plan de simetrie 2
R_e
R_i
Plan de simetrie 1
Grosimea g
h
b.
a
r
b
c
3r
x
m
n
y
p
c.
8
p > 0
Săgeta la montaj, Δ_m
F_m
Forma arcului montat (ambreiaj cuplat)
Forma arcului nemontat
d.
p = 0
Săgeata la decuplare, Δ_d
R_2
R_3
R_1
F_d
Forma arcului la decuplare
e.

A.2. Application description
Structure and operation of the clutch with diaphragm spring. The mechanical clutch in the figure above transmits the torque by friction from the flywheel assembly 1 and the pressure plate 3 to the disc 2 and through the groove to the main shaft of the gearbox 6. This process occurs when the lever 7 is inactive and the plate is activated. pressure 3 is pressed by the diaphragm spring 4 on the disc 2 and the steering wheel 1. At the action of the lever 7, the pressure bearing 5 presses on the diaphragm spring internally, removing the pressure disc 3 and interrupting the torque transmission. With the reduction of the pressing force on the pressure bearing the diaphragm spring returns (sometimes aided by another elastic element) and the coupling is performed. ***Assembly and operation of the diaphragm spring***. On the first hand, the diaphragm spring type element assures the function of generating the initial pressing force. It is mounted in the subassembly of the pressure disc 3 which is then mounted in the general assembly by means of screw assemblies 8 and, on the other part, of the displacement function of the pressure disk for decoupling. The first function involves the deformation of the outer part, similar to a disc spring with increased rigidity, by moving the area with radius R_2 with the arrow Δ_m reduced leading to an increased force F_m. The second function involves the deformation of the diaphragm spring, supported in the base area of the wings with reduced rigidity, under the action of the decoupling force F_d, it produces the decoupling by unloading the outer part of the press area, with the reduced value Δ_d.

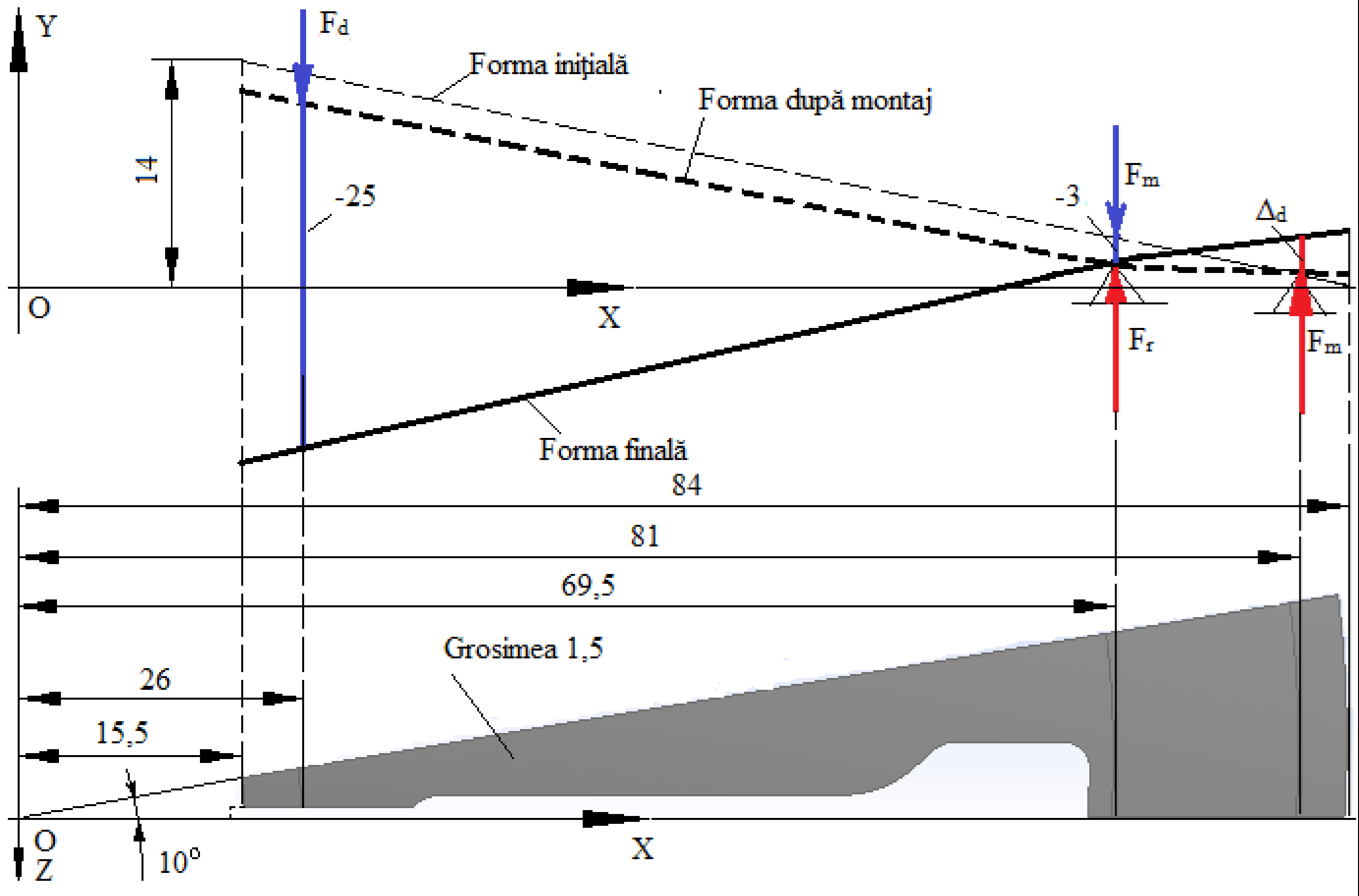
A.3. The application goal
In this application it is necessary to determine the force-displacement characteristics, the functional restrictions and the capable loads of the diaphragm spring in the figure above, considering that it is made of 50VCr11 spring steel and has the following dimensions: R_i = 15,5 mm, R_e = 84 mm, h = 14 mm, a = 1,5 mm, b = 3 mm, b = 10 mm, b = 3 mm, m = 3 mm, b = 28,5 mm, p = 10 mm, b = 3 mm, x = 0,75 mm, y = 3,5 mm, R_1 = 19 mm, R_2 = 68,5 mm, R_3 = 81 mm.

B. THE FEA MODEL

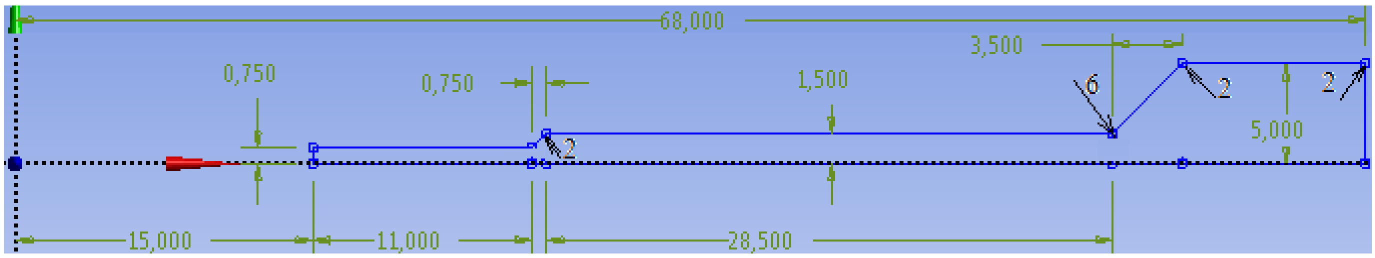
B.1. The model definition
Since the diaphragm spring has a reduced thickness (1.5 mm), the variations of the unknown internal parameters (displacements, deformations and stresses) are insignificant in the normal direction at the surface and a *2D model* is adopted for analysis. On the other hand, the structure of the spring being cyclically symmetrical circular is adopted for analysis only an angular segment (10°). Thus, without losing much of the accuracy, the problem to be solved falls into the state of *membrane tension* and a simplified possible model is adopted, which implies: • simple geometric shape, • adoption of material strength constraints (simple support), • geometrically nonlinear behavior with high imposed displacement loads, • linear behavior of the material.

B.2. The analysis model description
The geometrie of the analysis model is given by the surface of an angular sector (10°) to which the thickness of 1.5 mm is associated. For analysis the axial-symmetrical structure is modeled with <u>2D finite elements</u>. In order to simulate the behavior as close as possible to the reality, the two distinct functional states (assembly and decoupling) will be considered and consequently, the analysis will be done in two cases: the first implies a displacement imposed by the value - 2.5 mm of the points of the spring of circle with radius 69.5 mm (the bearing area on a toroidal ring) and the second, which over the previous loading also requires the movement of the points of the spring of 19 mm radius (action area) by -20 mm of the pressure bearing.

Thus, in the first model (mounting step) of analysis the structure will be supported simply (canceling the displacement in the direction of loading) on the pressure plate after the spring of 81 mm radius (action on the pressure disk) where the reaction F_m (unknown) appears) pressing on the pressure disc. In the case of the second model (decoupling), it will simply rest after the spring of the circle with the radius of 69.5 mm (the contact area with the other toroidal ring) where the reaction F_r (unknown) will appear; thus the outer part (deformed in the first stage) is relaxed and displaced by the value Δ_d (unknown) in the area of the spring with the radius 81 mm (contact area with the pressure disc) and in the contact area with the pressure bearing F_d (unknown).
In order to accurately highlight the functional processes for finite element analysis as a consequence of the geometric nonlinearity, the loads will be made progressively (the displacements imposed will be introduced in a table with the 1 mm step) and the Lagrange method will be adopted for solving.



a.



b.

B.3. Characteristics of the material and the environment

For linear static analysis the following resistance characteristics of 51VCr11 material are considered:

- longitudinal modulus of elasticity, E = 206000 N / mm^2;
- Poisson's ratio, ν = 0,3.

Average working temperature of the subassembly, T_0 = 20 ° C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project

WB, Toolbox: ↲⊟ Analysis Systems → ↲↲ Static Structural (the window with project modules appears automatically); [change name, Static Structural .....].

Setting of problem type (2D)

A : ↳ Geometry → ↲Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options, ↲ Analysis Type, [selecting from drop down list ↲▼, ↲2D] → [close the window ↲X].

Saving of the project

↲ Save As... → WB Save As, File name: [enter name, FEA-A.5] → ↲ Save

C.2 Modelling of material and environment characteristics

WB → Project Schematic → ↳ Engineering Data ✓ ◢ → ↲ Edit... → Outline of Schematic A2: Engineering Data: ↲ Structural Steel Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [selecting from drop down list C (Unit) cu / with ↲▼], [enter in column B (Unit) valoarea / value, 206000] → ↲ Update Project → ↲ Return to Project (others parameters are default).

C.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

WB, Project Schematic: ↳ Geometry → ↲DM New Geometry... → ANSYS Workbench: ↲ ⦿ Millimeter, ↲OK.

C.3.2 Sketch generation

Viewing default plane (XY)

DM, Tree Outline: ↲ Sketching → ↲ (Look at face/Plane/Schetch) → (automatically view of default plane XYPlane);

References lines generation

↲ Draw → ↲ Line → [the line will be drawn by selecting the first point with ↲, pe axa / on OX (the coincidence symbol appears, C), / moving to the other point (in the Ist cadran towards the OY axis) and release ↲, fig. a].

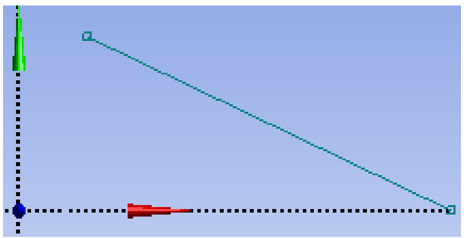
Dimensioning of reference line

↲ Dimensions → ↲ Display → [it is disabled with ↲ opțiunea / option / Name: / and activating Value:,
Name: ☐ Value: ☑];

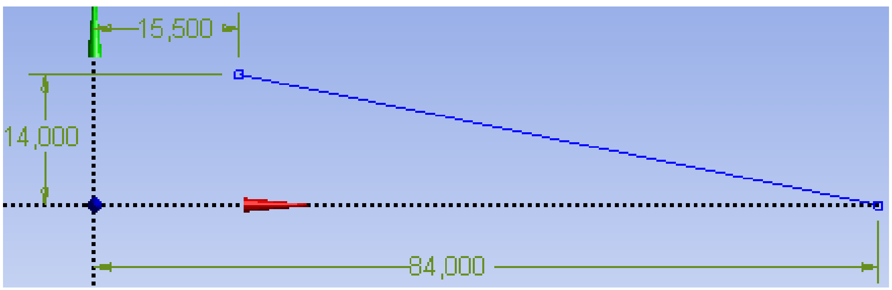
↲ Horizontal → [selecting succesive ↲ the pair: point on the reference line and the OY axis, the dimension appears (fig. b)] → Details View, Dimensions: 1 : ☐ H → [input value, 15,5/84,0].

↲ Vertical → [selecting with ↲ the point in Ist cadran and the axis OX, the dimension appears automatically (fig. b)] → Details View, Dimensions: 1 : ☐ V → [input value, 14,0].

↲ Move → [arranging the dimensions ↲ by draging to the desired position].



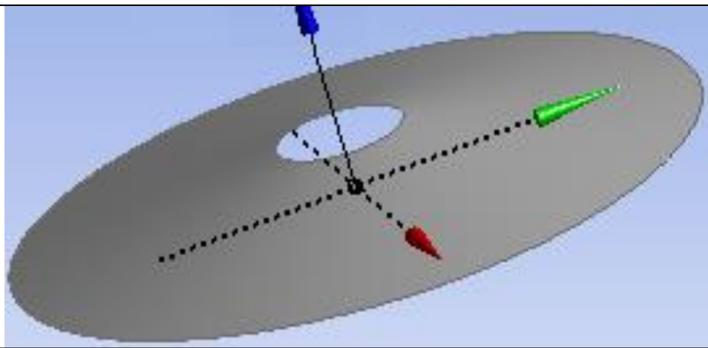
a.



b.

C.3.3 Surface generation

Tree Outline: ↵ Modeling ↵ Sketch1 ; ↵ Modeling → ↵ Revolve → Details View : Geometry → ↵ Apply →
↵ → [select with ↵ axe OY] → ↵ Axis → ↵ Apply ; ↵ Generate ; Tree Outline: ↵ ZXPlane ; : ↵
(visualization of the coordinate system attached to the geometric model).



C.3.4 Generating of helplines (for constraints and loading)

Generating sketch helplines

, Tree Outline: ↵ ZXPlane → ↵ Sketching → ↵ (New Sketch) [the name of the sketch is automatically indexed, Sketch2] ; ↵ (Look At Face/Plane/Sketch) [automatic view the selecting plane, ZX]; ↵ (view geometric model).

Generating helplines

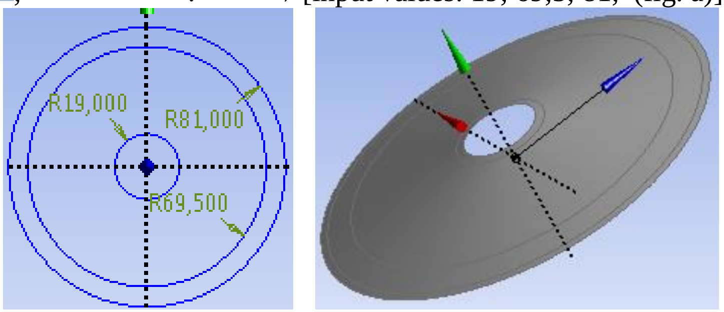
↵ Draw → ↵ Circle → [the circular line is generated by selecting with ↵ the center of the circle in the center of the coordinate system (coincidence symbol P appears), moving in radial direction and release ↵ on the contour, fig. a] (this sequence is performed three times for each circle).

Dimensions helplines

Tree Outline: ↵ Modeling → ↵ Sketch2 ; ↵ Dimensions → ↵ Radius → [select with ↵ the circular line, the dimension is automatically displayed] → Details View , Dimensions: 1 : R → [input values: 19; 69,5; 81, (fig. a)].

Printing helplines on the reference surface

: ↵ Extrude → ↵ Details View : ↵ Geometry → ↵ Sketch2 → ↵ Apply ; ↵ Direction Vector → ↵ Apply (the extrusion direction is accepted Normal defaulted to the line Direction), ↵ Operation → [set by default in the line with ↵ , Imprint Faces]; ↵ Generate .



a. b.

C.3.5 Generating the contour of the half-cut

Generating of half-cut sketch

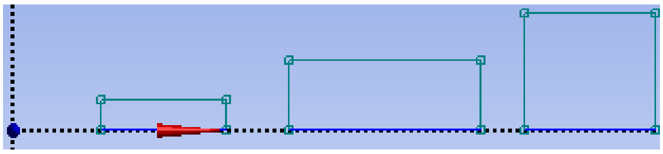
, Tree Outline: ↵ ZXPlane → ↵ Sketching → ↵ (New Sketch) [the sketch name is automatically indexed, Sketch3] → Details View , Details of Sketch3 : [change name Sketch3 name of the sketch, in Decupaj); ↵ (Look At Face/Plane/Sketch) [the selected plan will be automatically viewed, ZX]; ↵ (geometric model view); ↵ (geometric model view); Tree Outline: ↵ Sketch / Sketch2 → ↵ Hide Sketch). (the other sketches are masked).

Generating rectangular lines

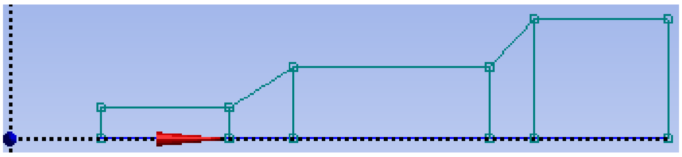
Tree Outline: ↵ Sketching → ↵ Draw → ↵ Rectangle → [select with ↵ the corner of the rectangular line at a point on the axis OX (coincidence symbol C appears), move the indicator and release ↵ in the other corner] (this sequence is carried out three times, fig. a).

Generating lines by two points

Sketching Toolboxes, Draw: ↵ Line → [select with ↵ pairs of points (coincidence symbol P appears)] (this sequence is reversed four times, including for the points on the X axis, fig. b).



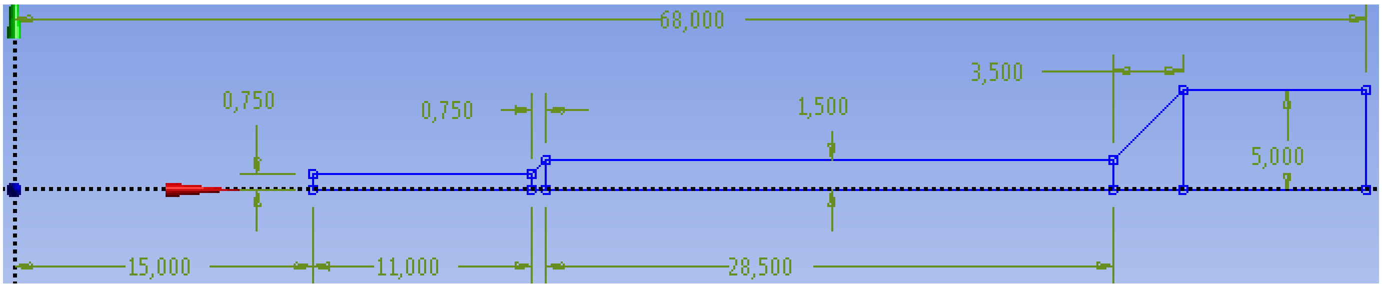
a.



b.

Dimensioning the sketch

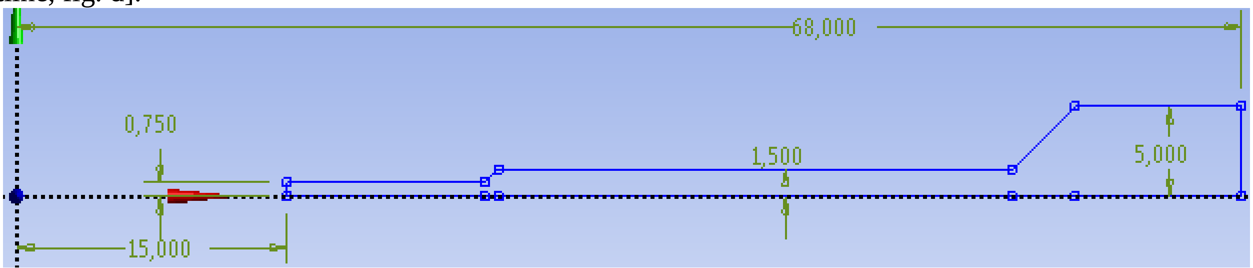
Sketching Toolboxes: ↵ Dimensions → ↵ Horizontal → [two entities (point, lines, axis) are selected with ↵ and the dimension appears automatically] → Details View, **Dimensions: 1**: ☐ H → [input value (fig. c)];
↵ Vertical → [two entities (lines) are selected with ↵ and the dimension appears automatically] → Details View, **Dimensions: 1**: ☐ V → [input value (fig. c)].



c.

Erasing lines

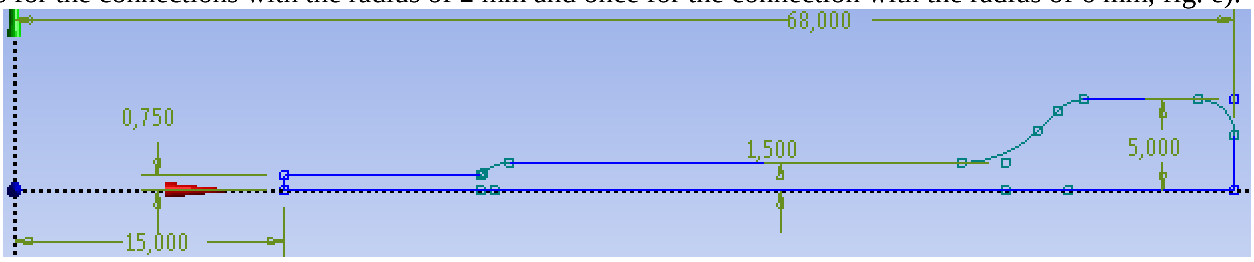
Sketching Toolboxes: →↵ Modify → ↵ Trim → [is deleted by selecting with ↵ lines that do not belong to the cut outline, fig. d].



d.

Generating radius

Sketching Toolboxes, Modify: →↵ Fillet → [input in Radius: value, 2/6] → [the intersection point of the two connected lines is selected with ↵ and the connection line is automatically displayed] (the sequence is repeated 3 times for the connections with the radius of 2 mm and once for the connection with the radius of 6 mm, fig. e).

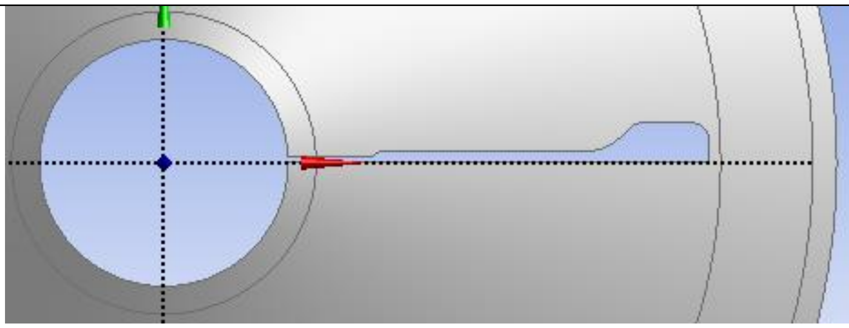


e.

C.3.6 Generating of half-cutting by material removal-

DM: ↵ Extrude → ↵ Details View
: ↵ Geometry → ↵ Decupaj
→↵ Apply,

↵Operation → [select with ↵ ▾, Cut Material] ↵ Apply (the default setting of the extrusion direction is accepted, Normal from line Direction); ↵ Generate



C.3.7 Model generation using cyclic symmetry

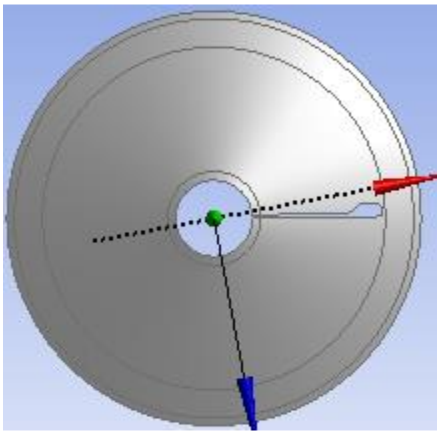
Generating of the symmetry coordinate system

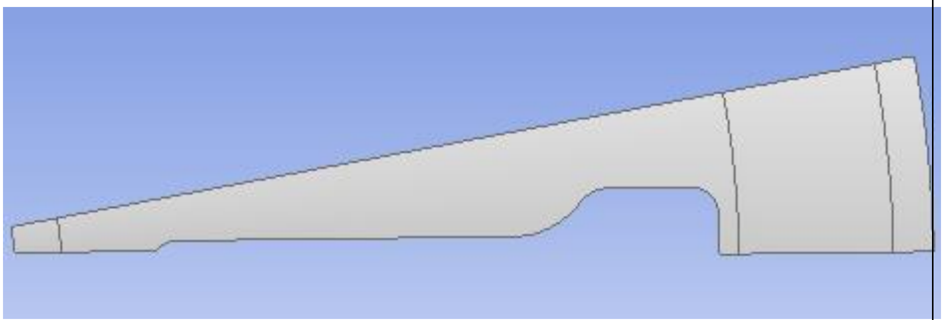
DM, Tree Outline: ↵ (New Plane) → ↵ XYPlane → Details View, Details of Plane4 : Type → [select with ↵ ▾, From Plane], FD1, Value 1 → [input value, -80], ↵Base Plane → ↵ Apply, Transform 1 (RMB) → select with ↵ ▾, Rotate about Y] (fig. a).

Model generation using symmetry

↵Tools → ↵ Symmetry → Details View, Details of Plane4 : ↵Number of Planes → [select with ↵ ▾, 2],

↵ YZPlane → Symmetry Plane1 → ↵ Apply, ↵ Plane4 → Symmetry Plane2 → ↵ Apply; ↵ Generate (fig. b);





a. ***b.***

C.3.8 Saving of geometric model

DM: ↵ (Save Project) → ↵ ✕ (Close Design Modeler).

C.4 Finite element modelling

C.4.1 Launching the finite element modelling module and setting the problem type, material characteristics, and unit system

Launching of the finite element modelling module

WB, Project Schematic: Model → ↵ M Edit... → [launch module *Mechanical [ANSYS Multiphysics]*.

Introduction of plate thickness and setting of material characteristics

↵ ⊞ Geometry → ↵ Surface Body → Details of "Surface Body", **Definition** : Thickness → [input value, 1,5 mm]; **Material** : Assignment , Structural Steel (usually, when there is only one material, this setting is default).

Setting the unit of measure system

M : ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) .

C.4.2 Model meshing

Setting global discretization parameters

M, Outline: ↵ Mesh, Details of "Mesh", ⊞ **Sizing** : Relevance Center → select from list with ↵ ▾, Medium / Fine

Automatically meshing

Mesh → ↵ Generate Mesh .

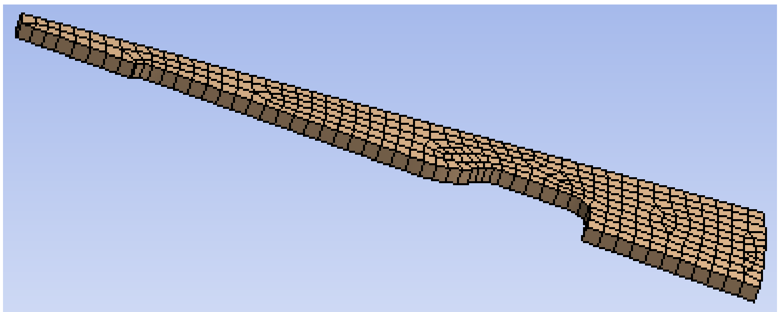
Setting the analysis parameters

Case I (assembly model)

Outline: ↵ Analysis Settings → ↵ Details of "Analysis Settings": ⊞ Step Controls: Number Of Steps → [input value, 3] (one step for each mm of deformation), ⊞ Solver Controls: Large Deflection → [select from list with ↵ , ↵ On] (geometric nonlinearity).

Case II (decoupling model)

Outline: ↵ Analysis Settings → ↵ Details of "Analysis Settings": ⊞ Step Controls: Number Of Steps → [input value, 28] (one step for each mm of deformation) ⊞ Solver Controls: Large Deflection → [select from list with ↵ , ↵ On] (geometric nonlinearity).

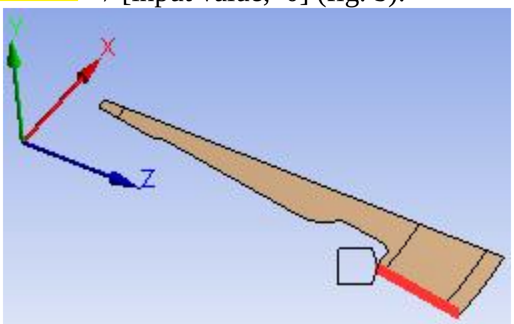


C.4.3 Supports and restraints modelling

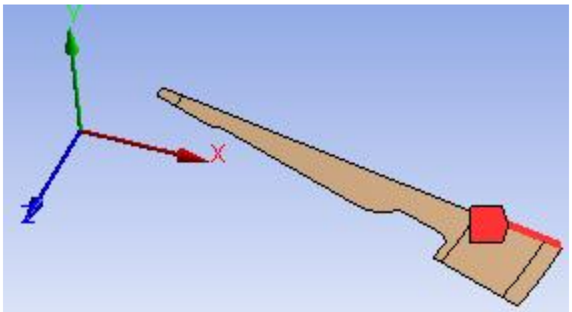
<u>Generating of tangential (circumferential) displacement constraints associated with radial lines</u> (ensures the behavior of the reduced symmetry model as well as the extended model)

, Outline: ↵ ⊟ Static Structural (A5) → ↵ Supports → ↵ Displacement ; → ↵ → [select with ↵+Ctrl of the radial lines in the XY (radial plane of symmetry) plane of the global coordinate system, fig. a]; Details of "Fixed Support": ⊟ Scope : ↵ Geometry → ↵ No Selection → ↵ Apply ; ⊞ Coordinate Systems → [select with ↵ , ↵ XYPlane]; Definition : ↵ X Component → [input value, 0] (fig. a).

↵ Static Structural (B5) → ↵ Supports → ↵ Displacement ; → ↵ → [select with ↵+Ctrl radial lines from plane Plane4 (the second simmetry plane, fig. b)]; Details of "Fixed Support": ⊟ Scope : ↵ Geometry → ↵ No Selection → ↵ Apply , ↵ Coordinate System → [select from list with ↵ , ↵ Plane4], Definition : ↵ Z Component → [input value, 0] (fig. b).



a.



b.

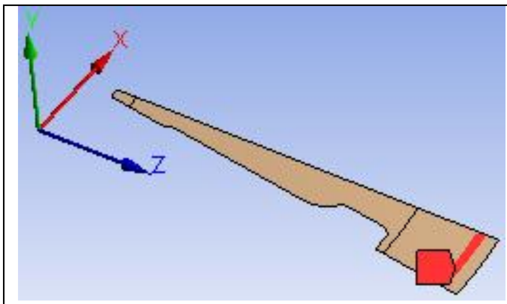
Case I (assembly model)

<u>Generating of constraints with zero displacements in axial direction</u> (in the bearing area on the pressure plate, fig. c)

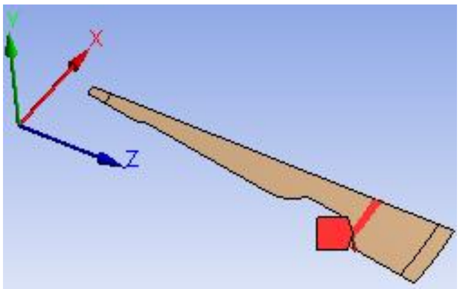
↵ ⊟ Static Structural (A5) → ↵ Supports → ↵ Displacement ; → ↵ → [selects the contact edge with the pressure disc]; Details of "Fixed Support": ⊟ Scope : ↵ Geometry → ↵ No Selection → ↵ Apply , Definition : ↵ Y Component → [input value, 0] (fig. c).

<u>Generating of constraints with imposed axial displacements</u> (in the resting area on the toroidal ring, fig. d)

↵ ⊟ Static Structural (A5) → ↵ Supports → ↵ Displacement ; → ↵ → [selection of the contact line with the toroidal ring, fig. d]; Details of "Fixed Support": ⊟ Scope : ↵ Geometry → ↵ No Selection → ↵ Apply , ↵ Y Component → [select from list with ↵ , ↵ Tabular] → Tabular Data → [input in the column ☑ Y [mm] values 0, -0,2, -0,4, -0,6] (fig. e).



c.



d.

Tabular Data

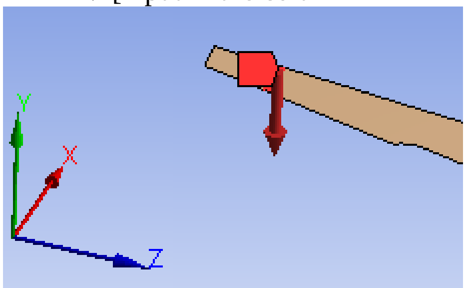
	Steps	Time [s]	☑ Y [mm]
1	1	0,	0,
2	1	1,	-0,2
3	2	2,	-0,4
4	3	3,	-0,6
*			

e.

Case II (decoupling model)

<u>Generating of constraints with imposed axial displacements</u> (in the contact area with the pressure bearing (fig. f)

↵ ⊟ Static Structural (A5) → ↵ Supports ▾ → ↵ Displacement ; → ↵ → [selection of the contact line with the pressure bearing, fig. f]; Details of "Fixed Support": ⊟ Scope : ↵Geometry → ↵ No Selection → ↵ Apply , **Definition** : ↵Coordinate System → [select from list with ↵▾, ↵Plane4], ↵ Y Component → [select from list with ↵ ▸, ↵Tabular] → Tabular Data: [input in the column ☑ Y [mm] valorile / values 0, -1, -2, …, -28] (fig. g).



Tabular Data

	Steps	Time [s]	☑ Y [mm]
1	1	0,	0,
2	1	1,	-1,
		. . .	
28	27	27,	-27,
29	28	28,	-28,

f. g.

Obs. In this case, the axial constraints imposed in case I remain active and the constraints with zero displacements in the axial direction from the bearing area on the pressure plate are deactivated (Outline: Displacement 3 → ↵ Suppress).

C.4.4 Loads modelling

Obs. Since the analysis with finite elements of this work is of a functional type (the deformed states are known in operation) and the loading forces are not known, the displacements imposed as constraints (see subchapter above) are considered as external loads with values. of unknown forces, to be determined as a result of this analysis.

D. SOLVING THE FEA MODEL

D.1 Setting the convergence criterion for solving the nonlinear geometric model

M, Outline: → ↵⊞ **Solution (A6)** → ↵ Solution Information , Details of "Solution Information",

↵⊞ **Solution Information**: ↵Solution Output → [selecting from the list with ↵▾, ↵Force Convergence] (the convergence of force is adopted).

D.2 Setting the results

<u>Selecting the total displacements</u>

M, Outline: ⊞ **Solution (A6)** → ↵ Insert → ↵Deformation → ↵ Total;

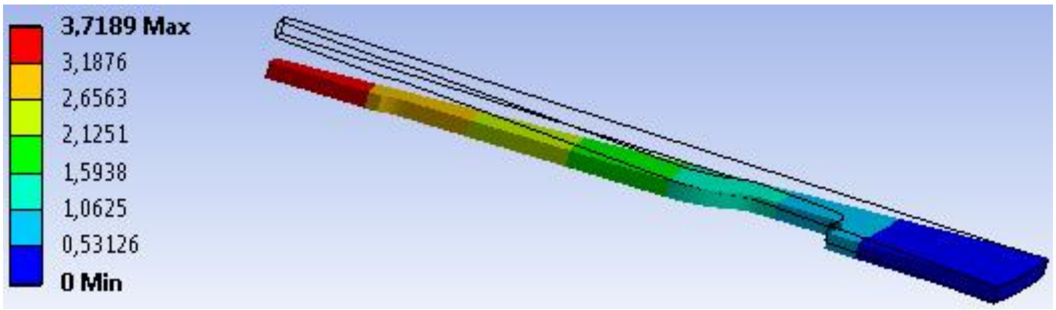
<u>Selecting the equivalent stress</u>

⊞ **Solution (A6)** → ↵ Insert → ↵ Stress → ↵ Equivalent (von-Mises).

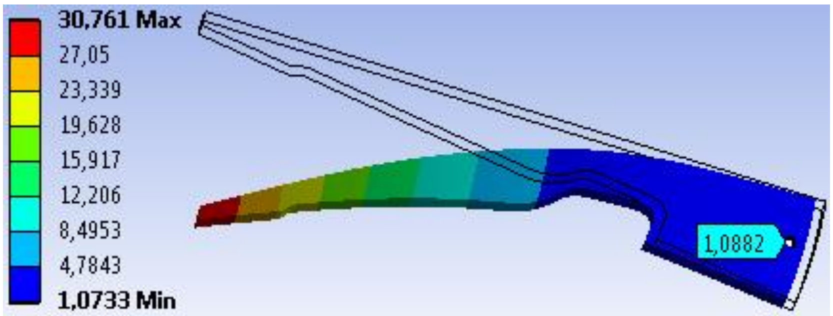
<u>Setting the circumferential stress</u> (in the normal direction on a plane of radial symmetry)

Generating of the cylindrical coordinate system: Outline: ↵ Coordinate Systems → Coordinate Systems: ↵ → Details of "Coordinate System": ↵ Definition: ↵ Type → [selecting from the list ↵, ↵ Cylindrical]; ↵ Origin → ↵ Define By [selecting from the list ↵, ↵ Global Coordinates]; ↵ Principal Axis: ↵ Axis → [selecting from the list ↵, ↵ Y]; ↵ Orientation About Principal Axis: ↵ Axis →[selecting from the list ↵, ↵ X].   *<u>Setting the normal stress along the Z axis of the generated cylindrical coordinate system</u>*   ↳ Solution (A6) → ↵ Insert → ↵ Stress → ↵ Normal → Details of "Normal Stress": ↵ Definition → ↵ Orientation → [selecting from the list ↵, ↵ Y Axis]; ↵ Coordinate System → [selecting from the list ↵, ↵ Coordinate System].   *<u>Setting the structural error</u>*   ↳ Solution (A6) → ↵ Insert → ↵ Stress → ↵ Stress → ↵ Error.   *<u>Reaction force setting</u> (in areas with imposed displacement)*   ↳ Solution (A6) → ↵ Insert → ↵ Probe → ↵ Force Reaction → Details of "Force Reaction": ↵ Definition → ↵ Boundary Condition, [selecting from the list ↵, ↵ Displacement]; ↵ Options → ↵ Result Selection → [selecting from the list ↵, ↵ Y Axis].
D.3 Launching the solving module
M, Outline: ↵ Solution (A6) → ↵ Solve.

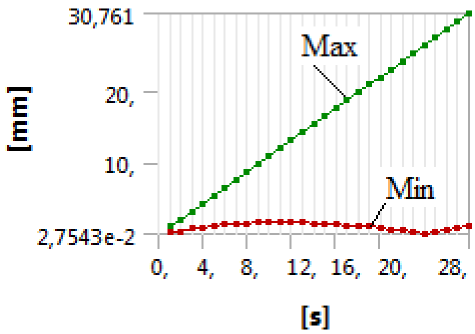
E. POST-PROCESSING OF RESULTS

E.1. Viewing the displacement field
M, Outline: ↵ Solution (A6) → ↵ Total Deformation.   ↵ ISO (axonometric view).   ↵ → [selecting from the list ↵, ↵ Smooth Contours] (visualization of smooth contours).   ↵ → [selecting from the list ↵, ↵ Show Undeformed WireFrame] (visualization of the non-deformed wireframe structure).   ↵ Result → [selecting from the list ↵, ↵ 1.0 (True Scale)] (selecting displacement scaling).   ↵ Probe → [is marked with ↵ a point in the area of the contact line with the pressure plate] (visualizing the value in a node).   Graph → ↵ Animation (animating view).      **Case I**

Case II



a.

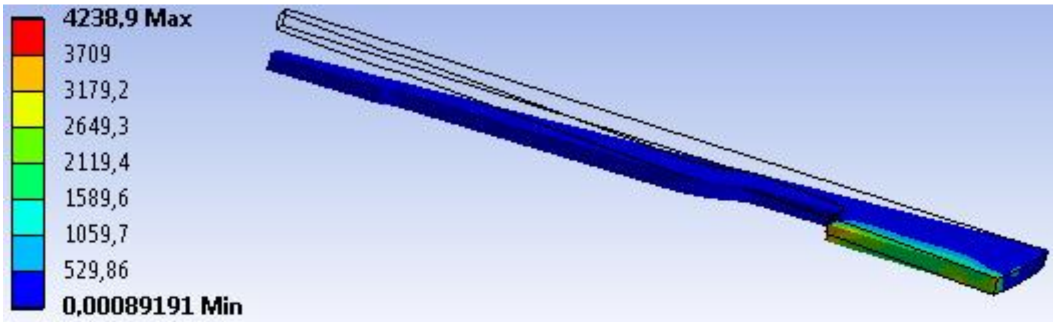


b.

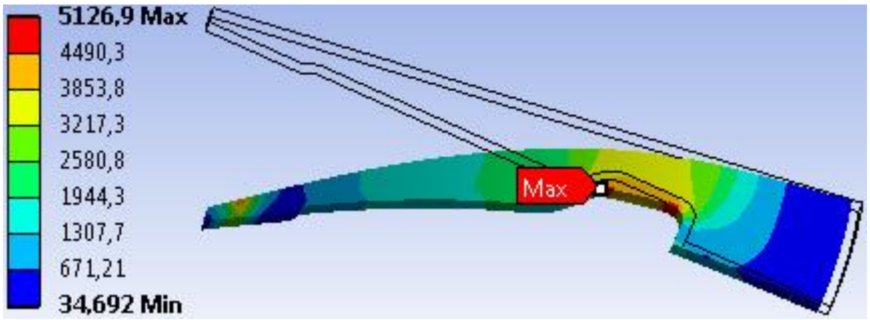
E.2 Viewing the equivalent stress field (von Mises)

Outline: ↵ ⊞ Solution (A6) → ↵ Equivalent Stress.

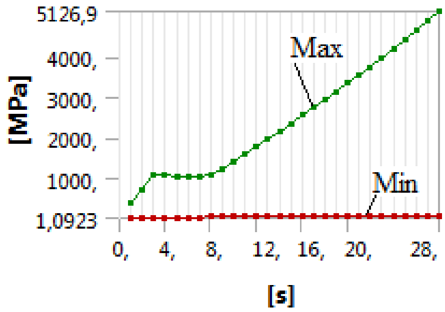
Case I



Case II



a.

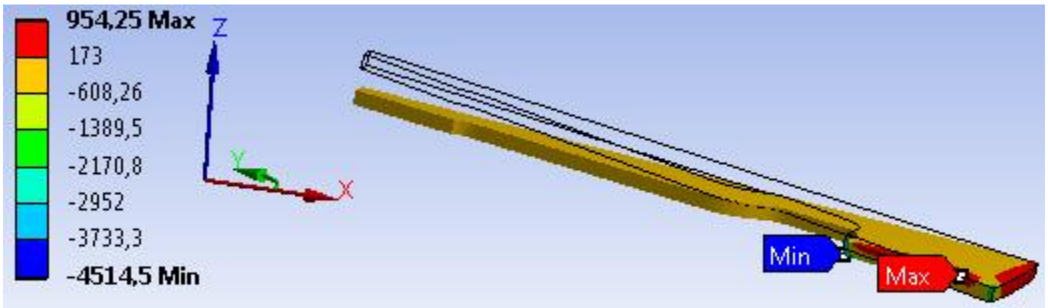


b.

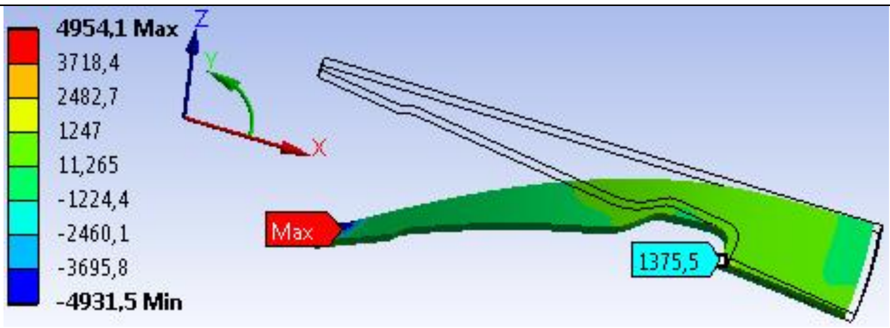
E.3 Viewing the circumferential stress field (radially normal)

Outline: ↵ ⊞ Solution (A6) → ↵ Normal Stress; ↵ MAX (measurement of the node with the maximum circumferential stress); ↵ MIN (measurement of the node with the minimum circumferential stress).

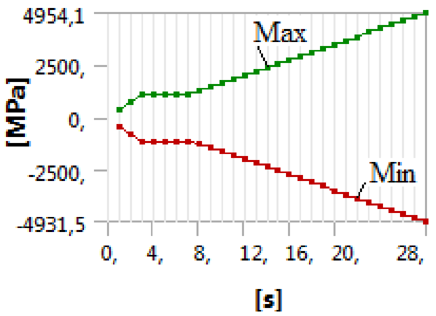
Case I



Case II



a.

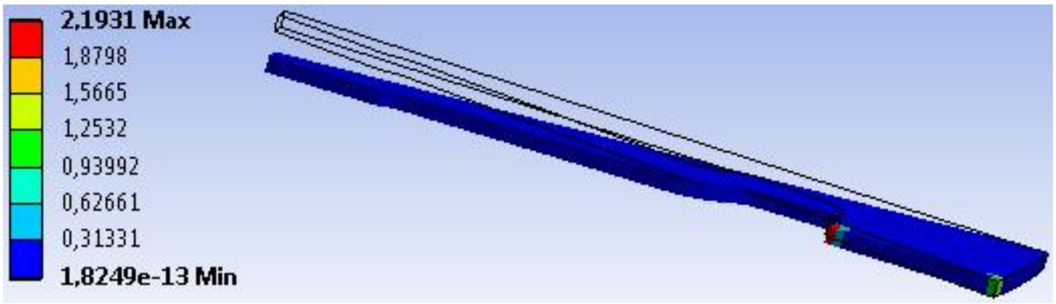


b.

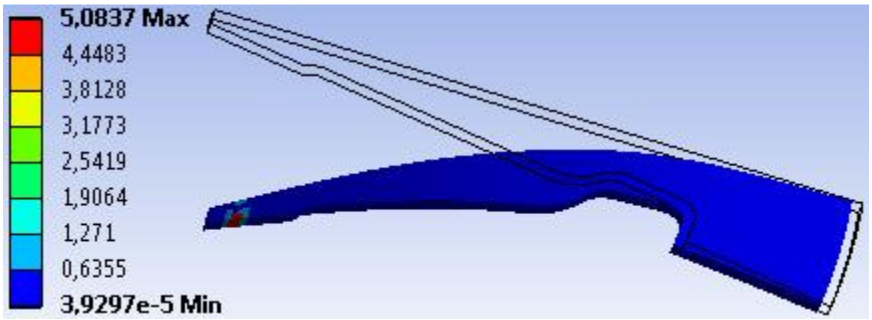
E.4 Visualization of the structural error field

Outline: ↵ Solution (A6) → ↵ Structural Error.

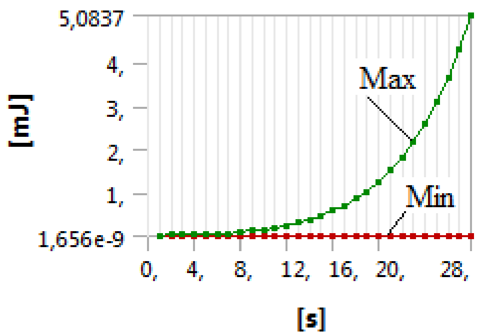
Case I



Case II



a.

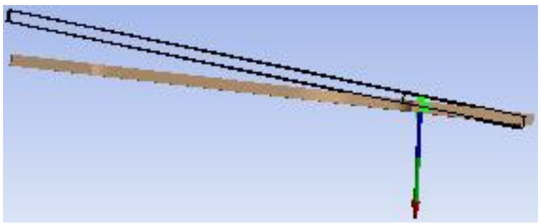


b.

E.5 Viewing the reaction forces

Outline: ↵ Solution (A6) → ↵ Force Reaction; [activation of the context menu with ↳ in the graphics area] → ↵View → Left (projection visualization, fig. a); ↵Graph → [the reaction force graph is automatically displayed according to the loading steps), fig. c]; ↵ Animation (viewing the animation).

Case I

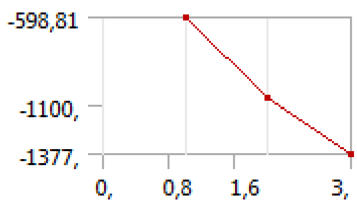


a.

Tabular Data

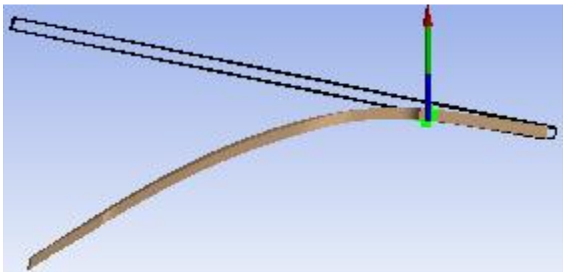
	Time [s]	Force Reaction (Y) [N]
1	1,	-598,81
2	2,	-1054,6
3	3,	-1377,

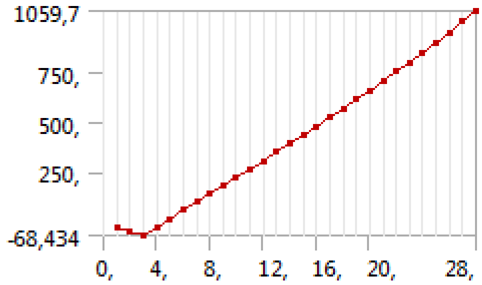
b.



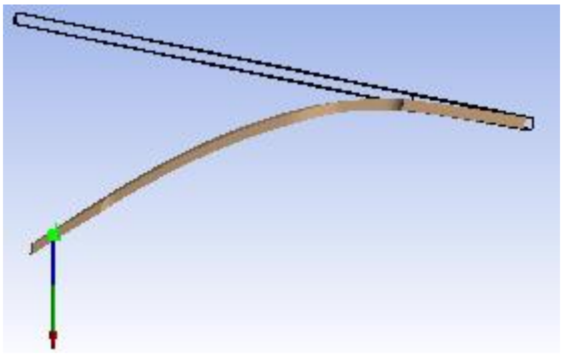
c.

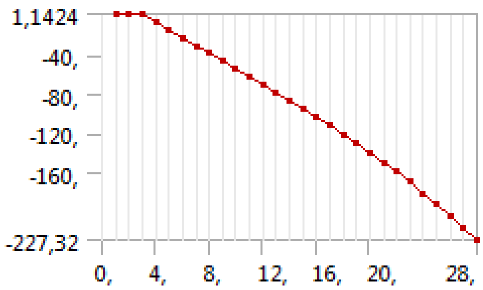
Case II





a. *b.*



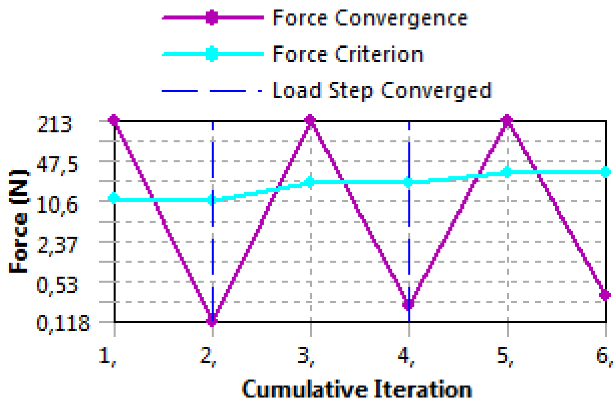


c. *d.*

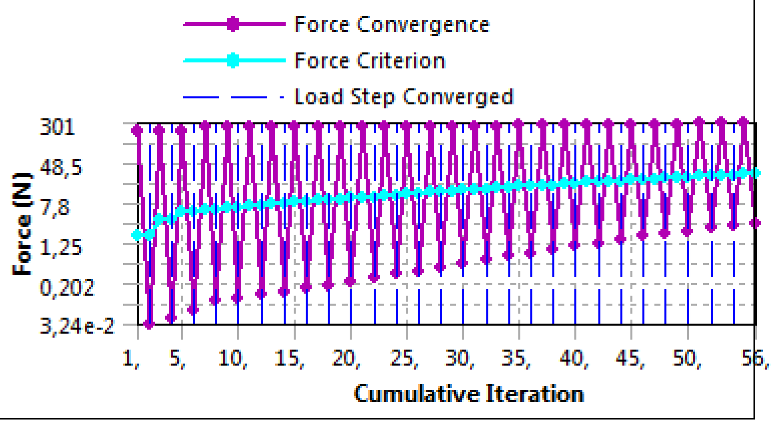
E.6 Viewing the convergence graphs of the solution of the nonlinear problem

Outline: Solution (A6) → Solution Information.

Cazul I / Case I



Cazul II / Case II



F. RESULTS ANALYSIS

F.1 Interpretarea rezultatelor / Interpretation of results
Following the analysis of the results obtained as a result of the modeling and FEA (subchapters E.1, E.2, E.3 and E.5) the following are highlighted:
Case I (fitting)
- The maximum total displacement (subchapter E.1, case I) of value 3.1709 mm from the area of the tip of the radial blade is generated by the deformation of the mounting disc; the radial blade remains undamaged.

- The maximum equivalent spring has the value 4238.9 MPa in the inner (compressed) area of the disc from the middle of the alveolus (subchapter E.2, case I); this value shows operation in the elasto-plastic field.
- Viewing the circumferential tension (normal on the radial plane; subchapter E.3, case I), shows positive and negative increased values (+ 954.25; -4514.5 MPa) in the outer (circumferential traction) and inner (respectively compression) areas circumferential); the maximum value in the compressed area the upper edge of the inner zone shows the operation in the elasto-plastic domain.
- The reaction force in the displacement zone imposed on the installation of -0.6 mm in three steps of -0.2 mm (incorrectly counted in time amounts [s]; subchapter E.4, case I, fig. B, c) has the maximum value, 1377 N; this value multiplied by double the number of blades determines the maximum pressing force of the pressure plate 3 on the disk 2 as a consequence of the tightening of the screws 8 (subchapter A.2, case I, fig. a, d, e); the value obtained is also used for calculating the threaded assemblies of the screws 8.

Case II (decoupling)

- As a result of the action of the pressure bearing and the elastic deformation of the radial blades it is observed that the outer area of the spring disk moves in the opposite direction with approx. 1 mm (subchapter E.1, case II, fig. A), the pressure plate is released and the decoupling occurs; in fig. b (subchapter E.1, case II) shows the variation of the maximum total displacement (green curve) and the variation of the minimum total displacement (red curve).
- The maximum equivalent stress has maximum values (<5126.9 MPa) in the areas of connection of the blade and of the action of the pressure bearing (subchapter E.2, case II, fig. A); these stresss appear as a consequence of the imposed displacement of the pressure bearing with non-real values (30 mm); for the real stroke (approx. 10… 15 mm) the maximum equivalent stress has values (approx. 2000 MPa, subchapter E.2, case II, fig. b) acceptable at design.
- The circumferential stresses on the upper and lower sides of the disc are quasi-horizontal (4954.1 MPa and 4931.5 MPa respectively; subchapter E.3, case II, fig.a, b); these values appear in the area of the displacement action line imposed on the pressure bearing which induces stress singularity; In the connection area of the blade to the disc there are very low stresss (<1500 MPa, subchapter E.3, case II, fig. a); consequence of the deformation of the blades in the disk the values of equivalent mounting stresss (subchapter E.2, fig. a) in the upper compressed area increase (approx. 1375.5 MPa).
- The reaction forces that appear in the zones with imposed displacements (in the bearing area on the toroidal rings (subchapter A.2, fig. A, d, e), when mounting and decoupling, and on the bearing on the pressure bearing, on decoupling) have the maximum values 1059.7 N and respectively - 227.32 N (subchapter E.5, case II, fig. a, c; for the design calculations of the debris subsystem, the pressure bearing and the pressure subassembly the values of these forces will be adopted according to the actual decoupling stroke (approx. 10… 12 mm, subchapter E.5, case II, fig.b, d).

F.2 Analysis of the precision and convergence of solving nonlinear models

Following the analysis of the obtained results, related to precision and convergence, as a result of the modeling and FEA (subchapters E.4 and E.6) the following are highlighted:

Case I (fitting)
- The maximum value of the structural error (2,1931mJ, subchapter E.4, case I) even in the area of maximum equivalent stress shows increased errors of its value; *in order to reduce errors, a more fine-grained rediscretion will be performed in this area and the analysis will be redone* - The convergence of the solution of the nonlinear model associated with the disk is made in 6 steps (subchapter E.6, case I); can be seen from fig. c (subchapter E.5) that the displacement force dependence is quasi-linear (the displacements are small).
Case II (decoupling)
- The structural error has the increased value (5.0837 mJ, subchapter E.4, a, b) in the action zone of the pressure bearing, modeled with imposed displacement associated to a line (theoretical situation), where much increased values of equivalent stress (*singularity of stress*); these values are not taken into account for the design; in order to avoid the singularity, the model is restored considering the *imposed displacement* associated with a contact surface or even considering *the direct contact between the blade and the pressure bearing ring* (situation very close to reality). - The convergence of the solution of the nonlinear model associated with the blade is made in 56 steps (subchapter E.6, case II); can be seen from fig. b (subchapter E.5, case II) that the displacement force dependence is nonlinear (the displacements are large).

G. CONCLUSIONS

In this paper, the modeling and analysis with finite elements were also done with didactic purpose following the *initiation of the user* with the main stages of development of an application of FEA in ANSYS Workbench, in which it is insisted, especially, on the modeling and analysis of a nonlinear elastic element diaphragm type with large displacements imposed.

The adopted FEA model has two superimposed functional states - assembly and decoupling with quasi-linear and respectively non-linear behaviors - and shows that in the action area (imposed displacement) of the pressure bearing increased values of the *structural error* (*stress singularity*).

As a result of solving the nonlinear model with finite elements adopting the force convergence method, results have been obtained with increased precision, the values of the obtained parameters (displacements, stresses, forces) being useful for designing the diaphragm elastic element as well as its neighboring elements within the clutch subassembly.

Application: FEA-A.6
Plastic deformation

KEY WORDS

Nonlinear static analysis, Spatial state of stresses, Nonlinear material, 3D geometric model, 3D finite element, Nonlinear finite element (parabolic), Cylindrical coordinate system, Mechanical contact without friction, Structural error, Plastic deformation Subset of processing

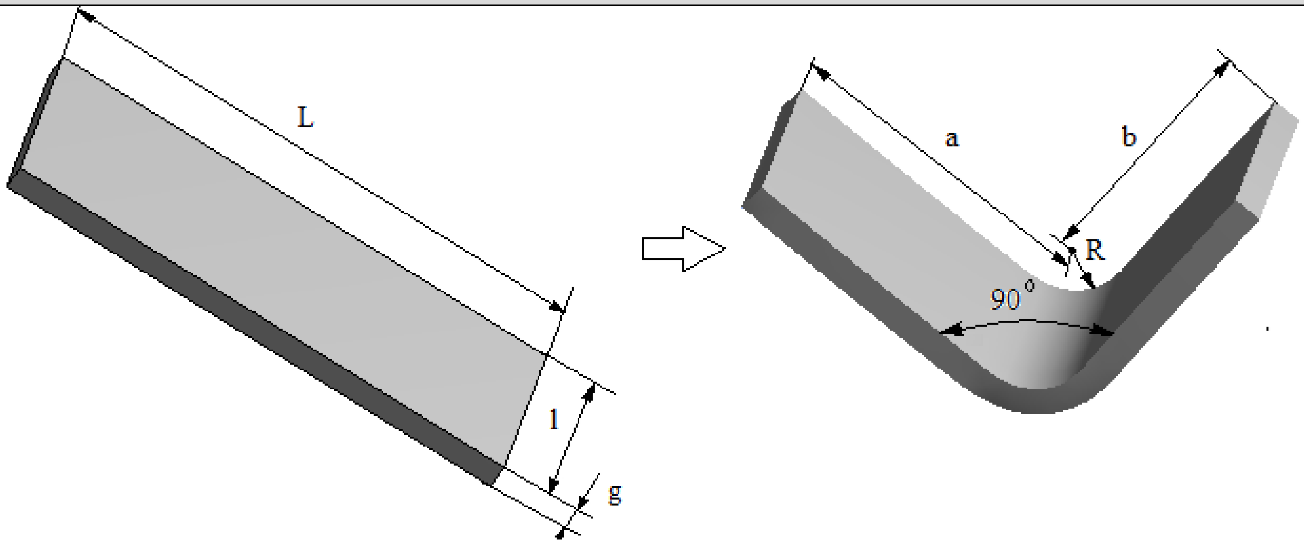
CONTENT

A. PROBLEM DESCRIPTION

A.1. Introduction

FEA, as a general method of studying physical phenomena and processes in mechanical structures also allows the analysis of the mechanical fields that appear in the case of *cold plastic deformation processes* of the thick sheets that assume the material parameters that describe the nonlinear *behavior with remaining deformations.*

A.2. Application description



The *cold bending* of the flatbed (the blank) in order to obtain the 90 ° corner piece with unequal wings implies the use of a die-punch device which involves fixing one wing and the plastic deformation of the other wing by means of the punch pressing it on the fixed die. After removing the punch, the piece remains in a deformed state. The material of the band is a soft (ductile) steel that involves *increased plastic deformation capacity* in interaction with the active parts of the device which are made of hardened non-plastic steel.

A.3. The application goal

This application assumes the FEA of the *bending process of a flat panel* with the length L = 105.7 mm, the width l = 40 mm and the thickness g = 5 mm in order to obtain a corner at 90o with uneven wings a = 40 mm and b = 50 mm. In the case of this application, it is necessary to establish *the maximum deformation load F* without having an excessive flow or the break established by the values of the maximum stresss that appear in the critical areas. In addition, following the analysis will be followed the determination of the values of the *pressures* in the interaction zones of the semi-manufactured with the active elements (die, punch) of the deformation device, necessary for its design.

B. THE FEA MODEL

B.1. The model definition

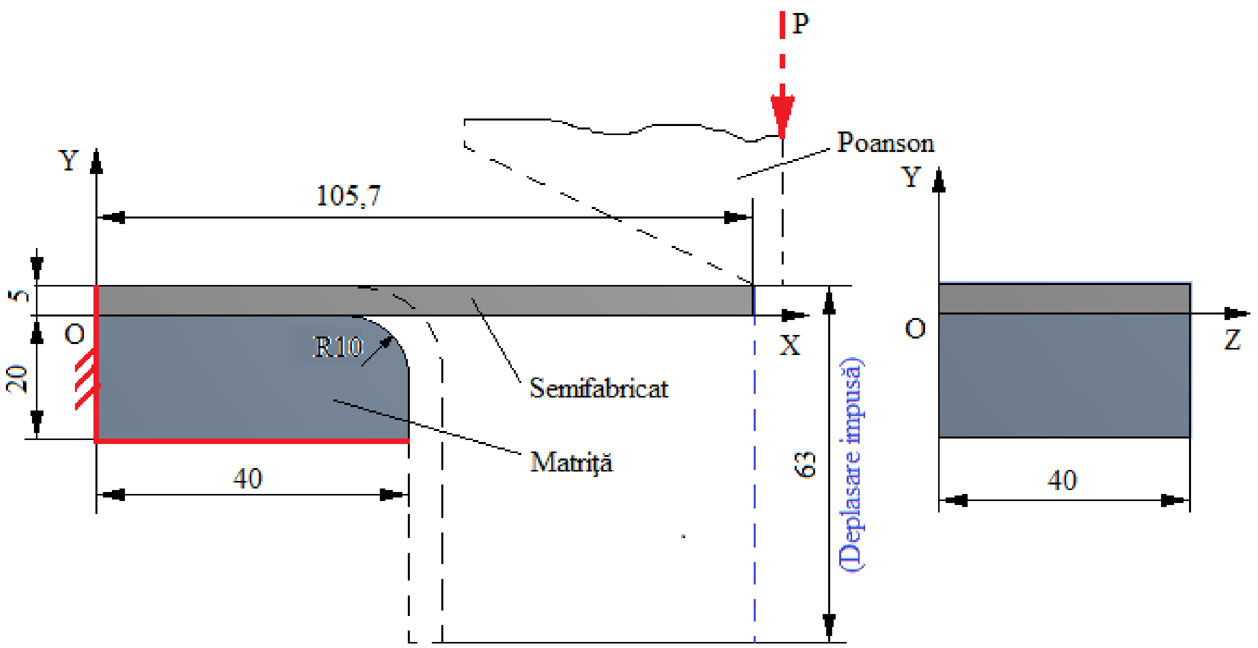
In order to design the FEA model, it is also necessary to consider the die-punch deformation device, adopting the following simplifying hypotheses:

- neglecting the effects of friction in mechanical contacts,
- adoption of material strength constraints (embedding, concentrated force action),
- the material has nonlinear elasto-plastic behavior according to a bilinear scheme.
- the deformation takes place static (the variation of the deformation force with time is not taken into account).

B.2. The analysis model description

The model for analysis is based on the 3D geometric model of the half-finished element in contact without friction with the 3D model of the active area of the mold. For analysis, the structure is composed of two solids that are modeled with 3D finite elements.

In order to simulate the plastic deformation as close to reality as possible, it will be necessary to move the edge of the half-finished element with the value -63 mm, in the direction of the axis of action of the punch. This constraint (displacement imposed) considered as an indirect load leads after the analysis to determine the value of the pressing force of the punch P.



B.3. Characteristics of the material and the environment
For the analysis with finite elements the strength characteristics of the materials are: • longitudinal modulus of elasticity E = 203000 N / mm2 (MPa), coefficient of transverse contraction (Poisson) ν = 0.29, modulus of plasticity Ep = 1800 MPa for steel of mechanical construction E295 (σ02 = 295MPa, σr = 490 ... 660 MPa) associated with the half-finished element. • the longitudinal elasticity modulus E = 210000 N / mm2, the coefficient of transverse contraction (Poisson) ν = 0.3, for the 40Cr10 alloy carbon steel (0.4% C and 1% Cr) associated with the die (Matrita) solid which, after the hardening treatment, reaches at hardness 50 ... 55 HRC. The average working temperature of the subassembly, $T_0 = 22^0$ C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project

WB, Toolbox: ↵⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural in *Def_pl*].

Problem type setting (2D)

A: ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from list with ↵▼, ↵ 3D] (this setting is usually default) → [close the window ↵ X].

Saving of the project

↵ Save As... → WB Save As, File name: [input name, FEA-A.6] → ↵ Save.

C.2 Modelling of material and environment characteristics

Generating of solid material characteristics Semifabricat

WB, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data:

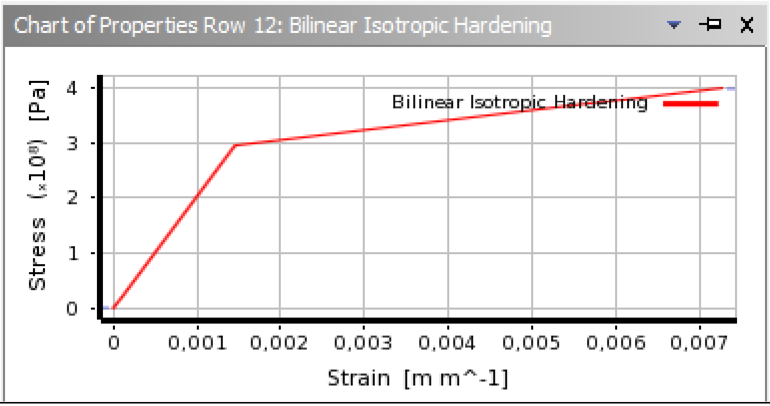
↵ Structural Steel → [the name will be changed to *Semifabricat*] (the features are set by default and the values will change); Toolbox: ↵ Bilinear Isotropic Hardening; Properties of Outline Row 3: Semifabricat: Young's Modulus → [select from list with ▼ MPa / input value: 203000], Poisson's Ratio → [input value:/ 0,29]; ? Bilinear Isotropic Hardening → Table of Properties Row 12: Bilinear Isotropic Hardening: Temperature (C) → [select from list with ↵ C (grade Celsius) / input value: 22], Yield Strength (MPa) ▼ → [select from list with ↵▼ MPa / input value 295], Tangent Modulus (Pa) ▼ → [select from list with ↵▼ MPa / input value 1800] (the window below is automatically generated).

Generating of solid material characteristics for the mold

Outline of Schematic A2: Engineering Data: ↵ Click here to add a new material → [the name of the *mold* is entered and the feature set appears,];, ? Matriță]; Toolbox:↵ Isotropic Elasticity; Table of Properties Row 2: Isotropic Elasticity:

Temperature (C) → [select from list with ↵ C (degree Celsius) / input value: 22], Young's Modulus (Pa) ▼ → [select from list with ↵ ▼ MPa / input value: 210000], Poisson's Ratio [input value: 0,3] (you can see the generation of these values as well as of others dependent on them and in the window Properties of Outline Row 4: Matriță). ↵

Update Project → ↵ Return to Project.



C.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

WB, Project Schematic: ↳ Geometry → ↵ DM New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.

C.3.2 Sketch generation

Viewing default plane (XY)

DM, Tree Outline: ↵ Sketching → ↵ (Look at face/Plane/Schetch) → (automatically view of default plane XY Plane);

Generating of rectangular contour Semifabricat

Sketching Toolboxes: ↵ Draw → ↵ Rectangle → [the rectangular line is generated in quadrant I, marking, with ↵, the first corner in the center of the coordinate system (coincidence symbol P appears) and the release ↵ in the opposite corner] (fig. a).

Dimensioning of outline sketch of Semifabricat

Sketching Toolboxes: ↵ Sketching → ↵ Dimensions → ↵ Semi-Automatic → [the semiautomatic dimensions are generated by marking with ↵); Display → [deactivate option Name: ☐ activate option Value: ☑] (the values of the dimensions on the drawing will be displayed); Details View, Dimensions: 2: ☐ L1 → [input value: 5], ☐ L2 → [input value: 40] (fig. a).

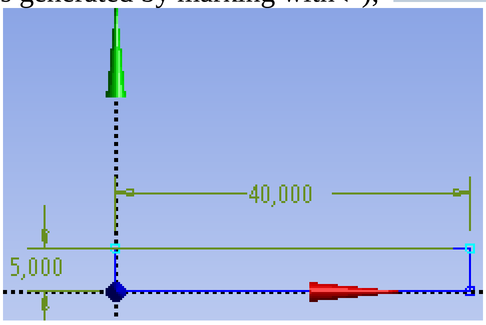
Generating of rectangular contour Matrita

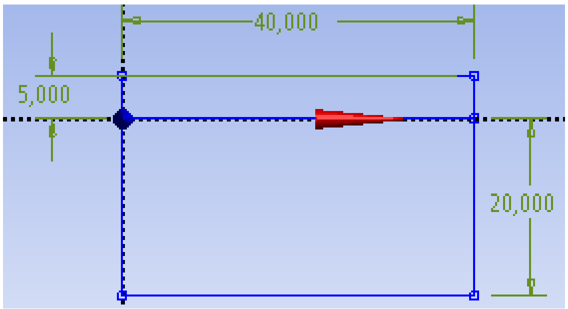
↵ Sketching → ↵ (New Sketch)→ (the sketch code is automatically indexed, Sketch2 ▾).

Sketching Toolboxes: ↵ Draw → ↵ Rectangle → [a rectangular line is generated with the side common to the previous sketch (coincident with the OX axis) by pressing ↵ the first corner in the lower left corner of the previous rectangle (coincidence symbol P appears and on overlapping with the axis OX symbol C) and releasing ↵ in the opposite corner] (fig. b).

Dimensioning the rectangular sketch of Matrita

↵ Sketching → Sketching Toolboxes: ↵ Dimensions → ↵ Semi-Automatic → [the dimension associated with the thickness is generated by marking with ↵); Details View, Dimensions: 1: ☐ L1 → [input value, 20] (fig. b).





a. **b.**

C.3.3 Generating the solids objects (Semifabricat, Matriţă)

Generating of solid Semifabricat

DM: ↵ Modeling → ↵ Sketch1; ↵ ISO (isometric view); ↵ (masking sketches);

↵ Extrude → Details View: Details of Extrude1, Geometry → ↵ Apply; ☐ FD1, Depth (>0) → [input value 105,7];

↵ Generate (fig. a).

↵ Solid → Details View, Details of Body, Body → [the default Solid name is changed to Semifabricat].

Generating of solid Matriţă

↵ Modeling → ↵ Sketch2;

↵ Extrude → Details View: Details of Extrude2, Geometry → ↵ Apply; ☐ FD1, Depth (>0) → [input value 50]; Operation → [select from list with ↵ ▾ Add Frozen] (solid separated from the previous one will be generated) ; ↵ Generate (fig. b).

↵ Solid → Details View, Details of Body, Body → [modify name: Solid în to Matriţă].

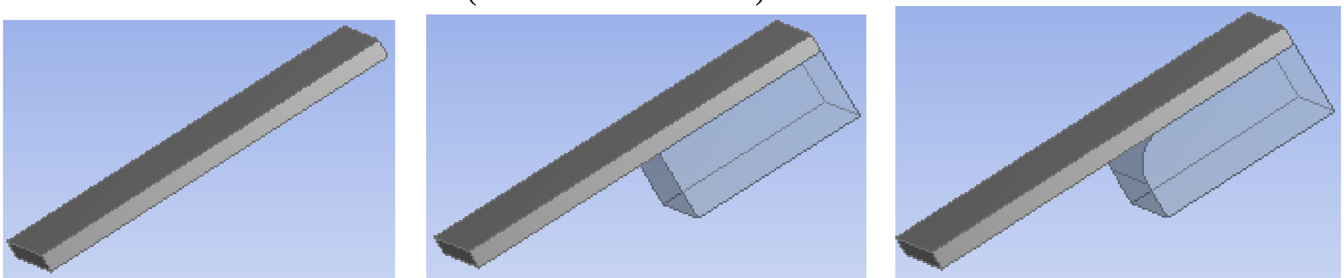
Generating of solid radius

↳ ✓ Semifabricat → ↲ Hide Body (hide the solid Semifabricat) ; Tree Outline: ↲ ✓ Matriță → ↲ → [it will be selected with ↲ the edge that will rotate] ;

↲ Blend → ↲ Fixed Radius; Details View, **Details of FBlend1**, FD1, Radius (>0) → [input value 10];

Geometry → ↲ Apply; ↲ Generate (fig. c).

↳ ✓ Semifabricat → ↲ Show Body (view solid Semifabricat).



a. *b.* *c.*

C.3.4 Saving of geometric model

: ↲ (Save Project) → ↲ (Close Design Modeler).

C.4. Finite element modelling

C.4.1 Launching of the finite element modeling module

Launching of the finite element modeling module

, Project Schematic: ↳ Model → ↲ **Edit...** → [launch modul *Mechanical [ANSYS Multiphysics*].

Setting the unit of measure system

: ↲ Units → ↲ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

Setting the material characteristics

, Outline: ↲ ⊞ ✓ Geometry → ↲? Semifabricat → Details of "Semifabricat", **Material** : ↲ Assignment → [select from list with ↲, ↲ Semifabricat] .

↲? Matriță → ↲ Details of "Matriță", **Material**, ↲ Assignment → [select from list with ↲, ↲ Matriță] .

Obs. In the specification tree, we observe, as a consequence of the connections between the two bodies, that a connection has been automatically generated in the subdivision ⊞ ✓ Connections o conexiune ✓ Contact Region, which will be further personalized.

C.4.2 Modeling the contact type

Generating of contact Semifabricat-Matriță

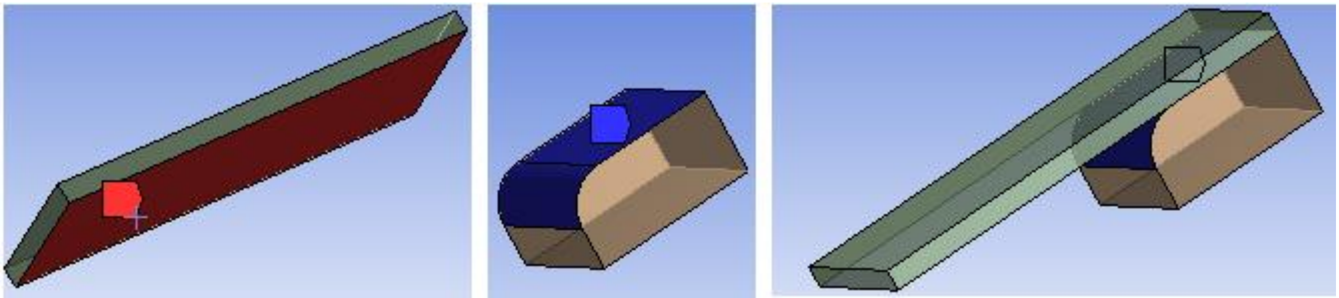
, Outline: ↲ ⊞ Connections → ↲ ✓ Contact Region → Details of "Contact Region", **Definition**: ↲ Type → [select from list with ↲, ↲ Frictionless].

Obs. If the initial contact generation command does not appear automatically, to initiate the contact ✓ Contact Region, the sequence is followed: ↲ ⊟ ✓ Connections → ↳ ✓ Contacts → ↲ Insert → ↲ Manual Contact Region, after which it is customized as above.

↲ ✓ Frictionless - Semifabricat To Matriță → ↳ ✗ Matriță → ↲ Hide Body (hide the solid Matriță) → ↲ →[is selected with ↲ the lower face of the entity Semifabricat, fig.a]→ Details of "Frictionless - Semifabricat To Matriță", **Scope**: ↲ Contact → ↲ Apply (option Contact Bodies indexing automatically, Semifabricat);

↳ ✗ Matriță → ↲ Show Body → ↳ ✗ Semifabricat → ↲ Hide Body (hide the solid Semifabricat) → ↲ → [select with ↲+Ctrl the initial contact seating face and the connecting surface, fig. b] → Details of "Frictionless - Semifabricat To Matriță", **Scope**: ↲ Target → ↲ Apply (option Target Bodies it is indexed automatically, Matriță); ↲ **Definition**: ↲ Behavior → [select with ↲, ↲ Symmetric]; ↲ **Advanced** → ↲ Formulation → [select with ↲, ↲ Augmented Lagrange] (method of solving the nonlinear model).

Obs. For a good convergence of the solution is adopted in the window at the Details of "Frictionless - option Target, in accordance with the Target entities (surfaces or edges) belonging to the fixed bodies, to the bodies with increased material rigidity (the longitudinal elasticity module may large) or have smaller curves.



a. ***b.*** ***c.***

C4.3 Setting discretization parameters, model discretization and analysis type setting

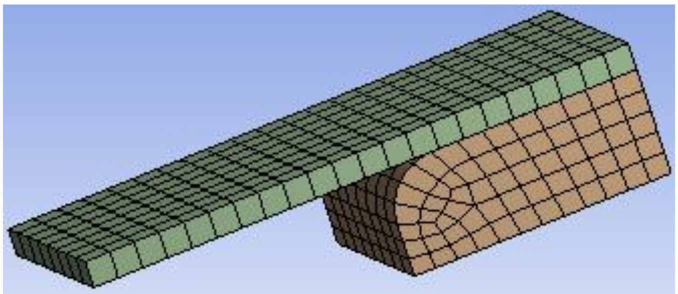
Setting the local discretization parameters in the contact areas

Outline: ↳ Mesh → ↵ Insert → ↵ Sizing; ↳ Matriţă → ↵ Hide Body → ↵ → [select with ↵ the lower face of the entity Semifabricat]; ↵ Sizing → Details of "Sizing" - Sizing Scope: ↵ Geometry → ↵ Apply; Definition → ↵ Element Size → ↵Default, [input value, 5].

↳ Mesh → ↵ Insert → ↵ Sizing; ↳ Matriţă → ↵ Show Body → ↳ Semifabricat → ↵ Hide Body → ↵ → [select with ↵+Ctrl the initial contact seating face and the connecting surface]; ↵ Sizing → Details of "Sizing" - Sizing: ⊟ Scope, ↵ Geometry, ↵ Apply; Definition: ↵ Element Size ↵Default, [input value, 5].

Automatic meshing

↳ Mesh → ↵ Generate Mesh.



Setting the analysis parameters

↵ Analysis Settings → Details of "Analysis Settings",

↵⊞ Step Controls: ↵Number Of Steps → [input value, 7];

↵⊟ Solver Controls: Large Deflection, → [select with ↵ , ↵On].

Obs. The displacements have large values and geometric type nonlinearity is adopted Large Deflection.

C.5 *Supports and restraints modelling*

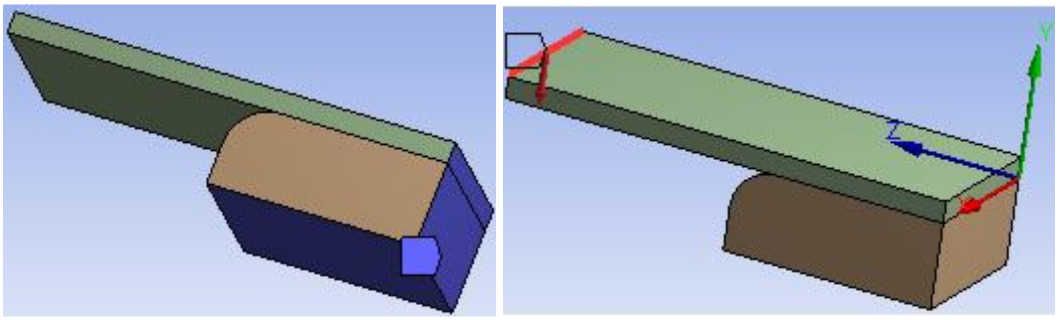
Generating of the constraint type (cancels all 6 degrees of mobility)

Outline: ↵⊞ Static Structural (A5) → ↵ Supports → ↵ Fixed Support; ↵⊞ Model (A4) → ↵ → [select with ↵+Ctrl faces with constraint]; ↵ Fixed Support → Details of "Fixed Support", Scope: ↵ Geometry → ↵ No Selection → ↵ Apply.

Generating of forced displacement constraint

↵⊞ Static Structural (A5) → ↵ Supports → ↵ Displacement; ↵⊞ Model (A4) → ↵ → [the edge of the entity Semifabricat on which the *Poanson* is pressed] ; ↵ Displacement → Details of "Displacement", Scope: ↵Geometry → ↵ No Selection → ↵ Apply; ↵Y Component → ↵ →

[select from list ↵, ↵Tabular] → Tabular Data → [input value in column Y [mm] valorile 0, -9, -18, … -63] (fig. c).



Tabular Data

	Steps	Time [s]	Y[mm]
1	1	0,	0,
2	1	1,	-9,
3	2	2,	-18,
	…		
7	6	6,	-54,
8	7	7,	-63,
*			

a. ***b.*** ***c.***

C.6. *Loads modelling*

Obs. Since the pressing force is unknown, it can be considered that the displacement imposed on it as a constraint (see subchapter above) is an external load of unknown value to be determined by this analysis.

D. SOLVING THE FEA MODEL

D.1 Setting the convergence criterion for solving the nonlinear physical model (with friction)

, Outline: → ↵ Solution (A6) → ↵ Solution Information , Details of "Solution Information",

↵ Solution Information : ↵Solution Output → [select from list with ↵, ↵Force Convergence] (the criterion of force convergence is adopted).

D.2 Setting the results

Selecting the total displacements

, Outline: Solution (A6) → ↵ Insert → ↵Deformation → ↵ Total.

Setting the displacement according to the Y direction

Solution (A6) → ↵ Insert → ↵Deformation → ↵ Directional; Details of "Directional Deformation", **Definition** :

↵Orientation → [select from list with ↵, ↵Y Axis];

Setting the equivalent stress

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Equivalent (von-Mises).

Setting the structural error

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Stress → ↵ Error.

Setting the reaction force *(in the displaced area)*

Solution (A6) → ↵ Insert → ↵ Probe → ↵ Force Reaction → Details of "Force Reaction":

↵ **Definition** → ↵ Boundary Condition, [select from list with ↵, ↵Displacement];

↵ **Options** → ↵ Result Selection → [select from list with ↵, ↵Y Axis].

Setting the parameters in the contact

Solution (A6) → ↵ Insert → ↵ Contact Tool → ↵ Contact Tool;

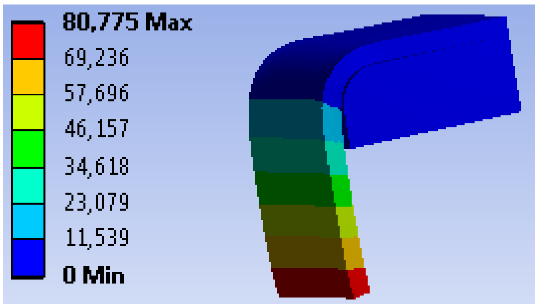
Contact Tool → ↵Insert → ↵ Status;

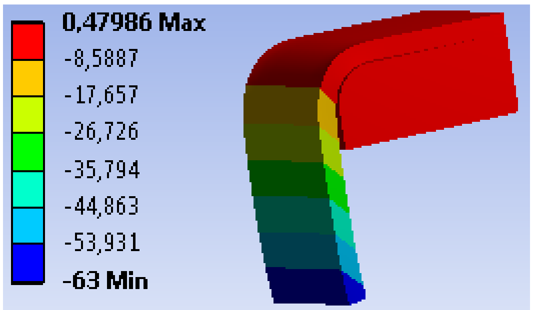
↳ Contact Tool → ↲ Insert → ↲ Pressure ;
↳ Contact Tool → ↲ Insert → ↲ Sliding Distance ;
↳ Contact Tool → ↲ Insert → ↲ Gap .
D.3 Launching the solving module
, Outline : ↲ Solution (A6) → ↲ Solve .

E. POST-PROCESSING OF RESULTS

E.1. Viewing the displacement field (total and Y axis)

, Outline: ↲ Solution (A6) → ↲ Total Deformation (fig. a); ↲ Directional Deformation (fig. b); ↲ ISO (axonometric visualization); ↲ → [select from list with ↲, ↲ Smooth Contours] (); ↲ → [select from list with ↲, ↲ Show Undeformed WireFrame] (visualization of the non-deformed wireframe structure). ↲ Result → [select from list with ↲, ↲ 1.0 (True Scale)] (selecting the scaling factor); Graph → ↲ Animation (view animation).

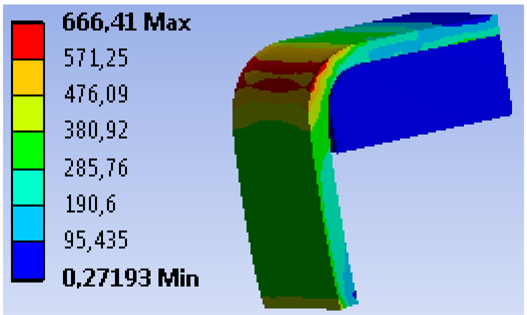


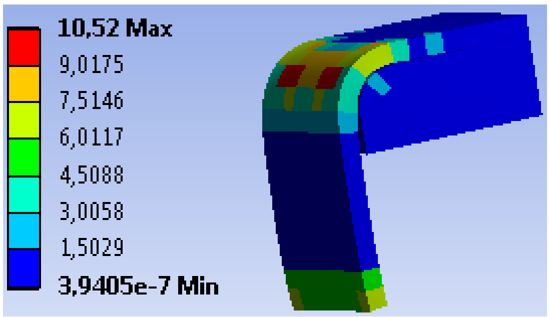


a. ***b.***

E.2 Viewing the equivalent stress field (von Mises) and structural error

, Outline: ↲ Solution (A6) → ↲ Equivalent Stress (fig. a); ↲ Structural Error (fig. b).

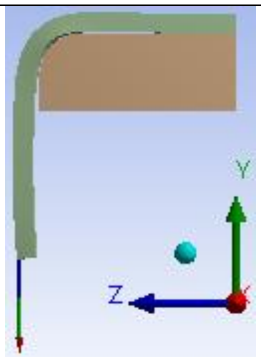




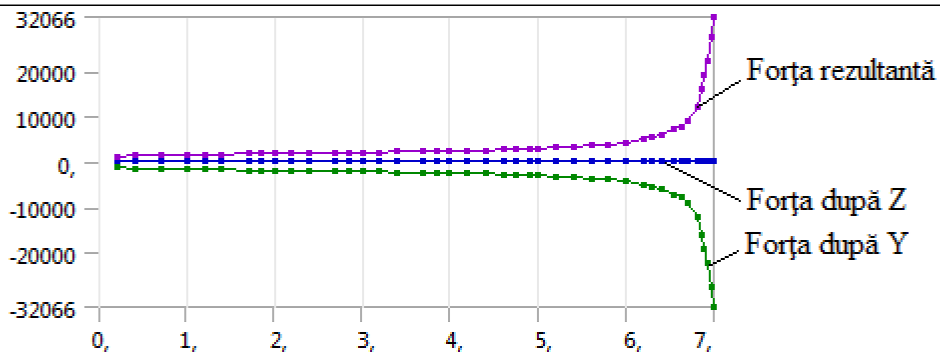
a. ***b.***

E.3. Visualization of the reaction force

, Outline: ↲ Solution (A6) → ↲ Force Reaction; [activation of the context menu with ↳ in the graphics area] → ↲ View → Right (projection visualization, fig. a); ↲ Graph → [the reaction force graph appears automatically according to the loading steps, fig. c]; ↲ Animation (view animation).



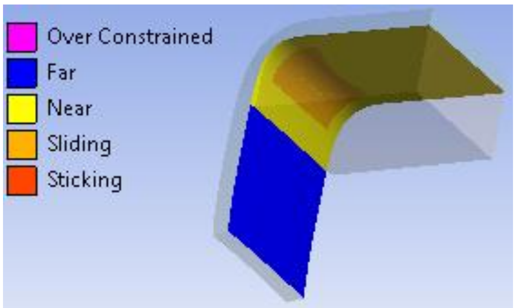
a.



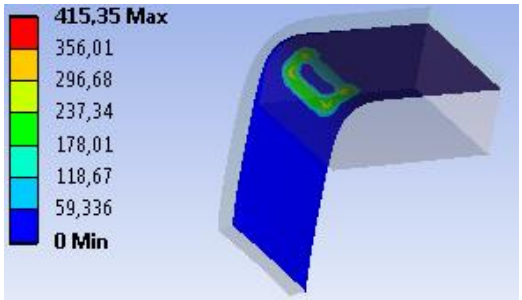
b.

E.4 Viewing the fields of the contact parameters (state, pressure, slip, jump)

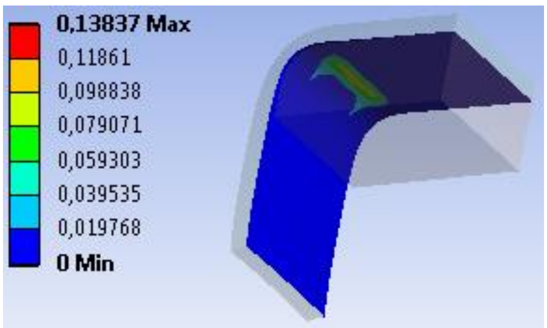
, Outline: ↵ ? Solution (A6) → ↵ Contact Tool → ↵ Status (fig. a); ↵ Pressure (fig. b); ↵ Sliding Distance (fig. c); ↵ Gap (fig. d).



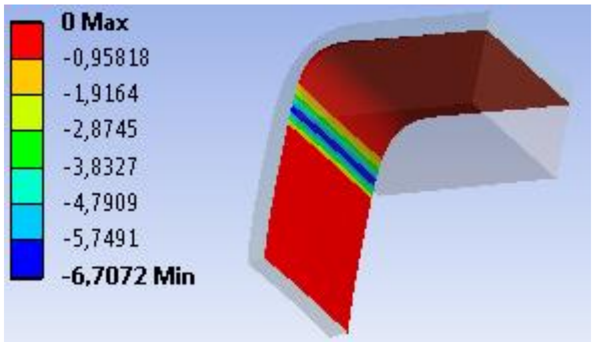
a.



b.

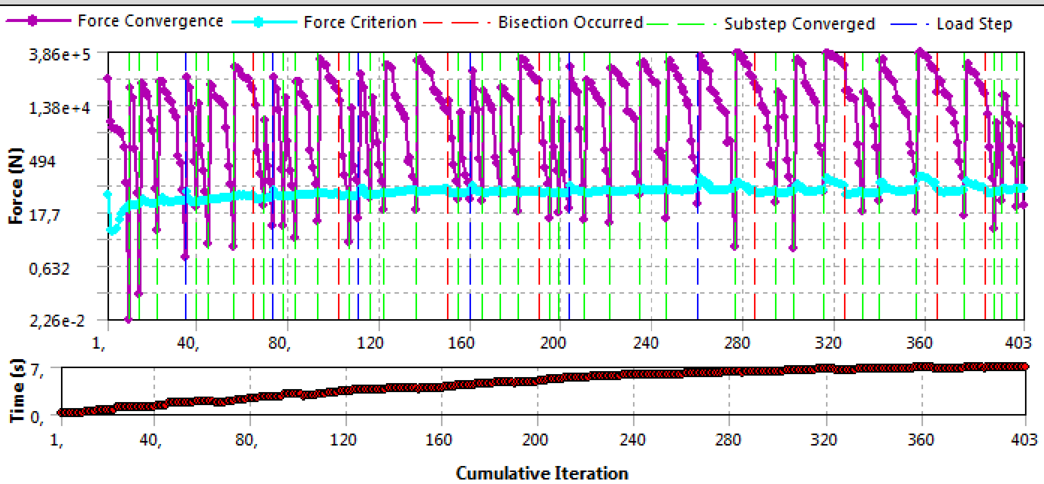


c.



d.

E.5 View the convergence graphs of the solution of the nonlinear problem



F. ANALYSIS OF RESULTS

F.1 Interpretation of results

Following the analysis of the results obtained as a result of the modeling and FEA (subchapter E) the following are highlighted:

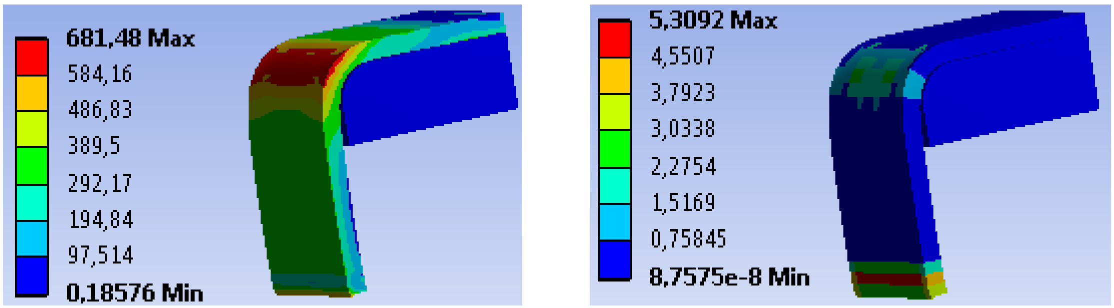
- Following the deformation process of the semi-finished product as a result of the action of the punch it is observed that the *wings are curved* (subchapter E.1); the maximum total displacement is 80,776 mm (subchapter E.1 fig.a); the displacement in the X-axis direction is 63 mm (subchapter E.1 fig.b), the same value imposed as constraint.
- The maximum equivalent stress has the value of 666.35 MPa (subchapter E.2, fig. A) in the outer curved area of the semi-manufactured greater than the flow tension (295 MPa, subchapter B.3) indicates the *existence of the plastic flow process*. On the other hand, the value of the maximum equivalent stress (666.35 MPa) being higher than the breaking stress of the material (max. 660 MPa, subchapter B.3) highlights the possibility of *breaking cracks* (subchapter F.3)
- The variation of the interaction force, increasing up to 32094 N, between the punch and the blank during the plastic deformation process is presented in subheading. E.3, fig. b. The values increased in the last part of the deformation process, situation shows that the value of the imposed displacement is greater than the real one and it is necessary to repeat the analysis with smaller values (eg 62.8 mm); the maximum value of the reaction force is the basis of the deformation device calculation.
- In the subcap. E.5 the contact states are visualized (subchapter E.5, fig. A) and the values of some contact parameters: pressure - max 415.35 MPa in the connection area, fig. b; relative slip - max 0.13837 mm in the upper area of the connection; play (jump) - max 6.7072 mm in the lower area of the connection. These values are useful for designing the workpiece and the mold. For example: starting from the maximum pressure value, the hardness of the active surface of the mold and the level of crushing of the semi-finished material inside the connection is determined; starting from the observation that the deformed wing of the blank is curved (subchapter E.4, fig. a; unwanted shape) and that the clearance between the die and the blank is increased (6.7072 mm) it is emphasized that the shape of the punch must be changed such as this to press on the semi-finished product and in the connection area a case involving the remodeling of the problem (subchapter F.3, fig.d)

F.2 Analysis of the precision and convergence of solving the nonlinear model

Following the analysis of the obtained results, related to precision and convergence, as a result, of the modeling and FEA (subchapters E.3 and E.6) the following are highlighted:

- The maximum structural error in the outer zone of the connection with increased value (10.52 mJ) shows increased deviations from the quasi-exact value (unknown). To reduce deviations, the fineness of discretization is increased and the model is solved. Thus, by changing the size of the finite element by scrolling through the sequence: ↲ ⊞ Mesh → ↲ Face Sizing / Face Sizing 2 → Details of "Face Sizing" - Sizing / Details of "Face Sizing 2" - Sizing, ↲ ⊞ **Definition**: ↲ ☐ Element Size → [enter the value, -3]. After solving (; ↲ Solve) the maximum reduced structural error, 5.3092 mJ, is obtained in the imposed displacement area (fig. B); the fact that in the area with the maximum equivalent stress (681, 48 MPa, fig. a) the values of the structural error are reduced (approx. 2… 3 mJ), it shows that the equivalent stress is very close to the quasi-exact one.

- The model solution convergence is done in 403 steps (subchapter E.6) and the computation time is increased.



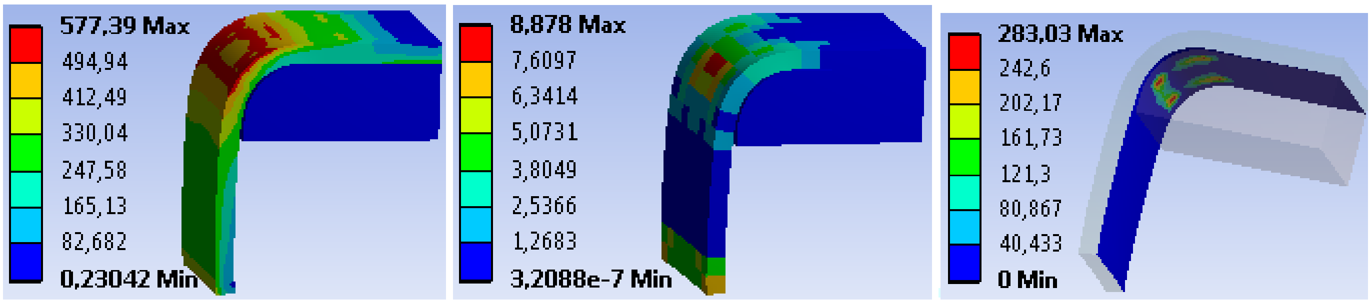
a. *b.*

F.3 Design studies

In order to avoid the occurrence of the rupture micro-cracks in the external connection area of the semi-finished product (subchapter F.1) it is necessary to reduce the maximum equivalent stress; In this case, the connection radius and / or the decrease of the plate thickness can be adopted, in compliance with the constructive-functional requirements. Thus, to increase the connection radius to the value of 15 mm, it is necessary to modify the analysis model and to solve the model by going through the sequences:

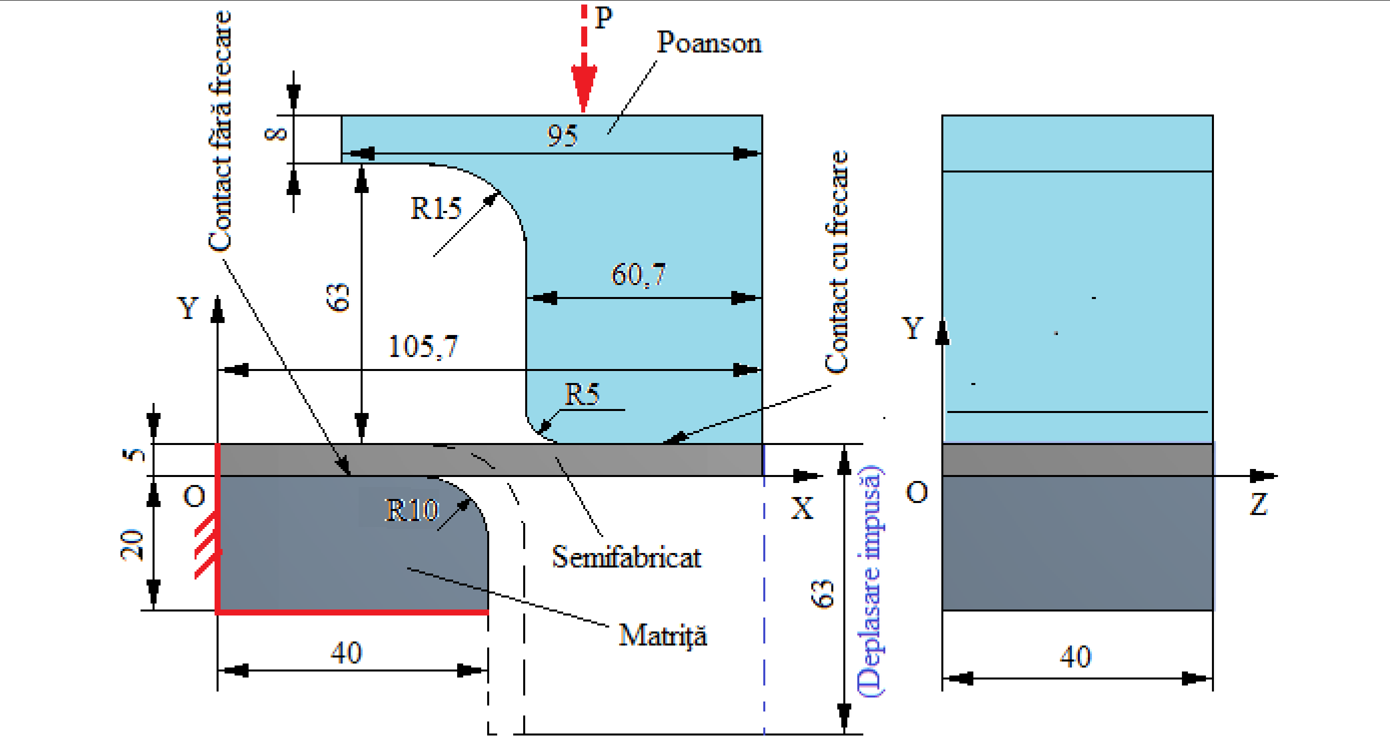
DM, Tree Outline: ↵ Extrude2 → ↵ FBlend1 → Details View, Details of FBlend1: ↵ FD1, Radius (>0) → [input value, -15]; ↵ Generate; M, Outline: ↳ Geometry → ↵ Refresh Geometry; ↵ Connections → ↵ Contacts → ↳ Contact Region → ↵ Delete (option Contact Region automatically apear with Refresh Geometry and because no other contacts are inserted, it is deleted); ↵ Displacement → Tabular Data: [change the value in step 7, -65 (instead of -63)]; ↵ Solve.

After solving, reduced values of the maximum equivalent stress (577.39 MPa, fig. a) and the contact pressure (283.03 MPa, fig. c) corresponding to the structural error of 8,878 mJ are obtained.



a. *b.* *c.*

If the design requirements require small deviations of the radius and linearity of the wings of the profile obtained from the imposed values, it is necessary that the bending device contain a punch with a contour that "forces" the plastic deformation of the blank to follow the contour of the mold (fig. d). Thus, the analysis model will have a third solid (Poanson) that will be in contact with the slip friction with the Semifabricat object (μ = 0.2). For FEA, as an exercise in this application, the same material as the mold will be adopted for the punch.



d.

G. CONCLUSIONS

In this paper, the modeling and the analysis with finite elements were also done with didactic purpose following the initiation of the user with the main stages of development of an application of FEA in ANSYS Workbench, which insists, especially, on the modeling and analysis of a deformable element in the plastic deformable area applying large displacements imposed.

The adopted FEA model involves considering the frictionless contact between two elements as well as a material with nonlinear behavior. The deformation force being unknown, the imposed displacement of the edge of the blank is introduced as loading.

As a result of solving the nonlinear model with finite elements adopting the method of force convergence, we obtained results with increased precision, the values of the obtained parameters (displacements, tensions, force) being useful for the design of the workpiece as well as of the bending device.

Application: FEA-A.7
Assembly on square profile

KEY WORDS
Non-linear static analysis, Spatial state of stresses, Linear material, 3D geometric model, 3D finite element, Non-linear finite element (parabolic), Cylindrical coordinate system, Mechanical contact with friction, Structural error, Assembly on square profile, Mechanical subassembly

CONTENT

A. PROBLEM DESCRIPTION

A.1. Introduction

FEA, as a general method of studying physical phenomena and processes in mechanical structures also allows the analysis of the mechanical fields that appear in the case of *the contacts of the mechanical assemblies* that suppose the elastic deformable surfaces from the direct contacts and of the slip friction that occur between them.

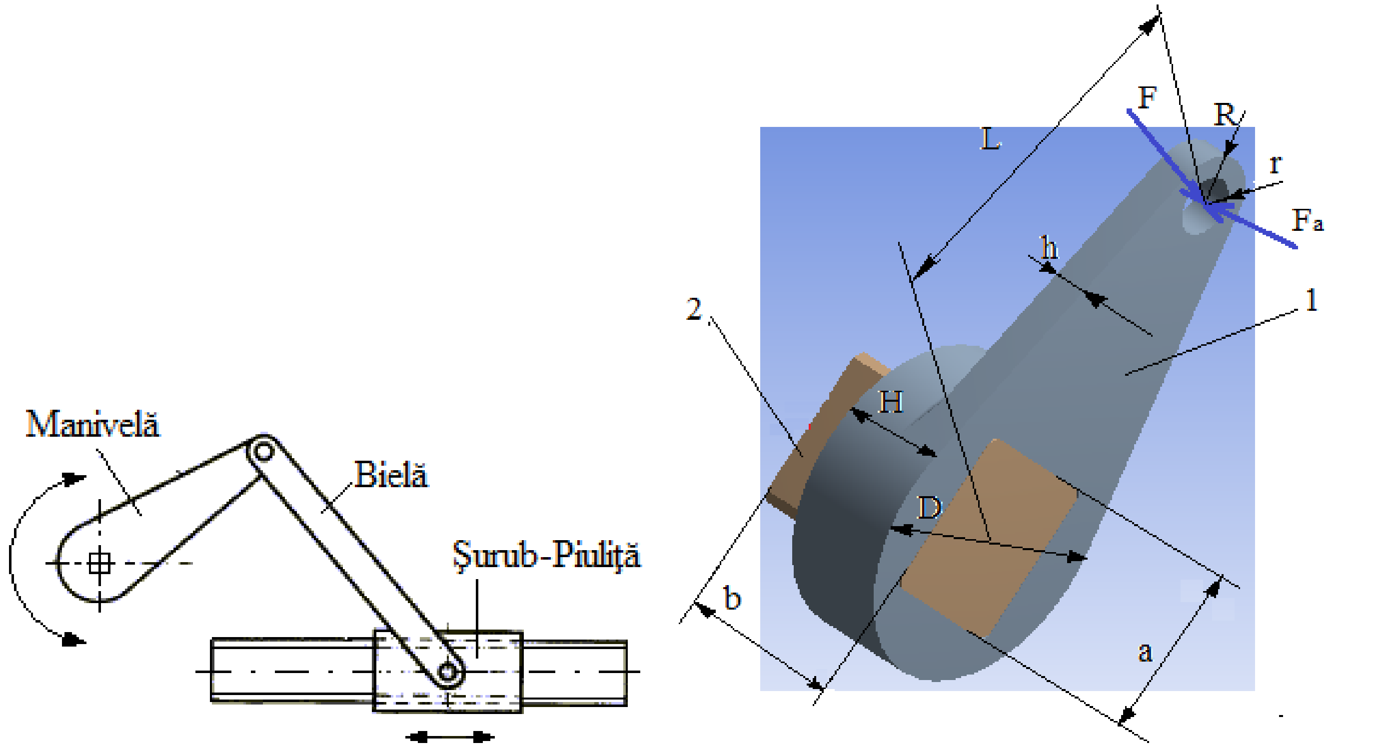
The profiled (polygonal) assemblies with the advantages of the increased load-carrying capacity and the good centering have disadvantages related to the very complex stress state in the contact areas, which requires modeling and FEA for design.

A.2. Application description

For the design of the crank of the mechanism that transforms the movement of swing of translation into movement of swing of rotation (fig. a) the head area of the driven shaft will be considered. The profiled (polygonal) square assembly of the figure *transmits the forces* from crank 1 to the square shaft 2 *by shape* (fig. b). For the design of the crank based on FEA, it is necessary to consider the interactions of its four internal contact surfaces with the four contact surfaces machined on the shaft. Although, the transmission of the forces from the crank to the shaft is done by form, in the four contacts during the elastic deformation of the materials in contact there appear relative litle spliping movements and therefore also frictional forces.

A.3. The application goal

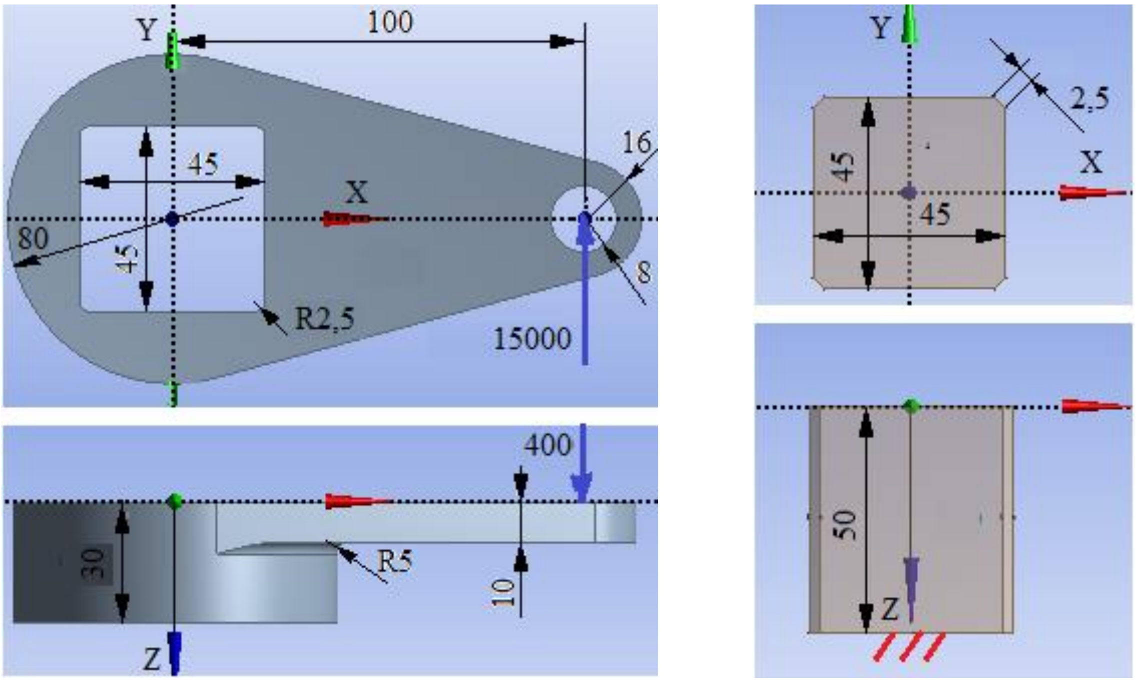
For this application, FEA is required for the displacement and stress fields in crank 1, including the shaft assembly with shaft 2. The crank is made of E335 soft steel and the improved C45 hard steel shaft. The dimensions of the assembly elements are: L = 100, h = 10 mm, H = 30 mm, a = 45 mm, b = 50 mm, R = 16 mm, r = 8 mm. The crank is loaded with tangential force F = 10000 N and axial F_a = 2000 N.



a. b .

B. THE FEA MODEL

B.1. The model definition
In order to design the crank FEA model, it is necessary to consider and a portion of the driven shaft adopting the following simplifying hypotheses: • considering friction in mechanical contacts, • adoption of material strength constraints (embedding, concentrated force action), • the material has elastic linear behavior, • the deformation occurs static (the variation of the deformation force with time is not taken into account).
B.2. The analysis model description
The model for analysis is based on the 3D geometric model of the crank (fig. a) in contact with friction with the 3D model of the shaft (fig. b). For analysis the structure is composed of two solids (crank and shaft) which are generated by extrusion and discretized with 3D finite elements. In order to make the *mechanical contacts between the crank and the shaft* on flat surfaces, the square profile of the crank will be connected to the corners with 2.5 mm radius and the shaft one will be 2.5 mm (fig. A, b). In order to simulate the behavior of the assembly as close to reality when modeling the mechanical contacts between elastic deformable surfaces, the friction coefficient μ = 0.2 will be considered. The loading of the model for the analysis will be carried out in the area of the crankshaft coupling (subchapter A.2, fig. a) with the tangential forces, 20000 N, and axial 4000 N (fig. a, b). The modeling of the loads with these forces will be done with the specific function "Bearing Load".



a. *b.*

B.3. Characteristics of the material and the environment

For the analysis with finite elements the strength characteristics of the materials are:

- longitudinal modulus of elasticity E = 205000 N / mm^2 (MPa), coefficient of transverse contraction (Poisson) ν = 0.29, for steel E335 (σ_{02} = 335MPa – at traction; σ_{02} = 400 MPa – at compresion; σ_r = 590…760 MPa) associated with Manivelă element.
- the longitudinal elasticity modulus E = 210000 N / mm2, the coefficient of transverse contraction (Poisson) ν = 0.3, for the C45 alloy carbon steel (0.4% C) associated with the Arbore solid which, after the hardening treatment, reaches at hardness 250 ... 280 HB (σ_{02} = 520MPa – at traction; σ_{02} = 560 MPa – at compresion; σ_r = 690…860 MPa)

The average working temperature of the subassembly, $T_0 = 22^0$ C.

C. PREPROCESSING OF FEA MODEL

C.1 Activarea şi salvarea proiectului / Creating, setting and saving the project

Creating of the project

WB, Toolbox: ↵⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural in *Assem_sq*].

Problem type setting (2D)

A : ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from list with ↵▾, ↵ 3D] (this setting is usually default) → [close the window ↵ X].

Save of the project

↵ Save As… → Save As, File name: [input name, FEA-A.7] → ↵ Save.

C.2 Modelling of material and environment characteristics

Generating of the material characteristics for the component Manivelă

WB, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Click here to add a new material → [input name: E335] (it apear in line ? E335) → ↵? E335 → Toolbox: ↵⊞ Linear Elastic

→ ↵↵ Isotropic Elasticity → Table of Properties Row 2: Isotropic Elasticity: Temperature (C) → select from list with ↵ C (grade Celsius) input value, 20], Young's Modulus (Pa) → [select from list with ↵ MPa, input value 205000], Poisson's Ratio → [input value 0,29] (the window is automatically generated Properties of Outline Row 4: E335 and the graph Chart of Properties Row 2: Isotropic Elasticity, in which the data entered are highlighted).

Generating of limit and permissible characteristics of the material E335

, Toolbox: ↵ ⊞ Strength → Tensile Yield Strength → Properties of Outline Row 4: E335: ? Tensile Yield Strength, [select from list with ↵ MPa input value, 335] (traction limit stress).

Toolbox: ↵ ⊞ Strength → Compressive Yield Strength → Properties of Outline Row 4: E335: ? Compressive Yield Strength, [select from list with ↵ MPa, input value, 400] (compresion limit stress).

Toolbox: ↵ ⊞ Strength → Tensile Ultimate Strength → Properties of Outline Row 4: E335: ? Tensile Ultimate Strength, [select from list with ↵ MPa / input value, 220] (traction admisible stress).

Toolbox: ↵ ⊞ Strength → Compressive Ultimate Strength → Properties of Outline Row 4: E335: ? Compressive Ultimate Strength, [select from list with ↵ MPa input value, 300] (compresion admisible stress).

Generating of the material characteristics for the component Arbore

, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Click here to add a new material → [input name C45] (it apear in line 4 ? C45) → ↵ ? C45 → Toolbox: ↵ ⊞ Linear Elastic → ↵↵ Isotropic Elasticity → Table of Properties Row 2: Isotropic Elasticity: Temperature (C) → [select from list with ↵ C (grade Celsius) input value, 20], Young's Modulus (Pa) → [select from list with ↵ MPa / input value, 210000], Poisson's Ratio → [input value 0,3] (the window is automatically generated Properties of Outline Row 4: E335 and the graph Chart of Properties Row 2: Isotropic Elasticity, in which the data entered are highlighted).

Generating of limit and permissible characteristics of the material C45

, Toolbox: ↵ ⊞ Strength → Tensile Yield Strength → Properties of Outline Row 5: C45: ? Tensile Yield Strength, [select from list with ↵ MPa / input value, 520] (/ traction limit stress).

Toolbox: ↵ ⊞ Strength → Compressive Yield Strength → Properties of Outline Row 5: C45: ? Compressive Yield Strength, [select from list with ↵ MPa / input value, 560] (compresion limit stress).

Toolbox: ↵ ⊞ Strength → Tensile Ultimate Strength → Properties of Outline Row 5: C45: ? Tensile Ultimate Strength, [select from list with ↵ MPa / input value, 420] (traction admisible stress).

Toolbox: ↵ ⊞ Strength → Compressive Ultimate Strength → Properties of Outline Row 5: C45: ? Compressive Ultimate Strength, [select from list with ↵ MPa / input value, 480] (compresion limit stress).

Update of the database and exit

: ↵ Update Project → ↵ Return to Project.

C.3. Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

, Project Schematic: ↳ Geometry → ↵ New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.

C.3.2 Sketch 1 generation (Shaft)

Viewing default plane (XY)

, Tree Outline: ↵ Sketching → ↵ (Look at face/Plane/Schetch) → (automatically view of default plane XY Plane);

Generating of rectangular contour

Sketching Toolboxes: ↵ Draw → ↵ Rectangle → [a rectangular line is generated in the center area of the XY plane, marking with ↵ the first (quadrant II) corner and the release ↵ in the opposite corner (quadrant IV)] (fig. a).

Centering of rectangular contour

Sketching Toolboxes: ↵ Constraints → ↵ Symmetry → [select with ↵ the Y axis followed by the select selection of the two lines parallel to the Y axis (the two sides will be positioned centered relative to the Y axis)] (fig. b) → [is selected with ↳ a dot in the model area] → ↵ Select new symmetry axis → [select with ↵ the X axis followed by the selection with ↵ of the two lines parallel to X (the two sides will be centered centered with X)] (fig. b).

Dimensioning of rectangular contour

Sketching Toolboxes: ↵ Dimensions → Horizontal → [select with ↵ lines parallel to the Y axis] (the dimension is automatically viewed) → Details View, Dimensions: , ☐ H → [input value, 45] (fig. c);

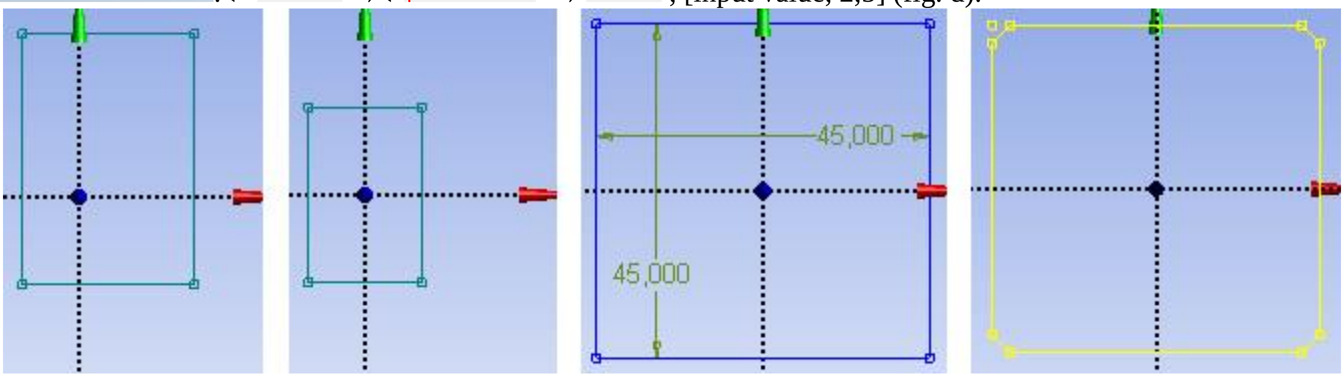
↵ Vertical → [select with ↵ liniile paralele cu X / paralle lines with X] (the dimension is automatically viewed) → Details View, Dimensions: 2 : ↵☐ V → , [input value, 45] (fig. c);

Changing the dimension view

↵ Display → ↵Name: ☑ (desactivate) → ↵ Name: ☐ Value: ☑ (activate, fig. c); ↵ Move → [select with ↵ and drag in wanted position] (fig. c).

Generating the chamfers

Sketching Toolboxes: ↵ Modify → ↵ Chamfer → Length: , [input value, 2,5] (fig. d).



a. ***b.*** ***c.*** ***d.***

C.3.3 *Generating of sketch 2 (crank / hub)*

Generating sketch 2

↵Sketching → ↵ (New Sketch)→ (the sketch code is automatically indexed, Sketch2).

Generating overlapping lines over Sketch1

↵Sketching → ↵ Line → [select with ↵ the end points of the line respecting the coincidence condition P that appears when overlapping them] (the line appears automatically after selecting the second point; this sequence is repeated four times, fig. a); ↳ Sketch1 → ↵ Hide Sketch (hide Sketch); ⊞ XYPlane → ↵ Sketch2 (activate Sketch).

Generating the chamfers

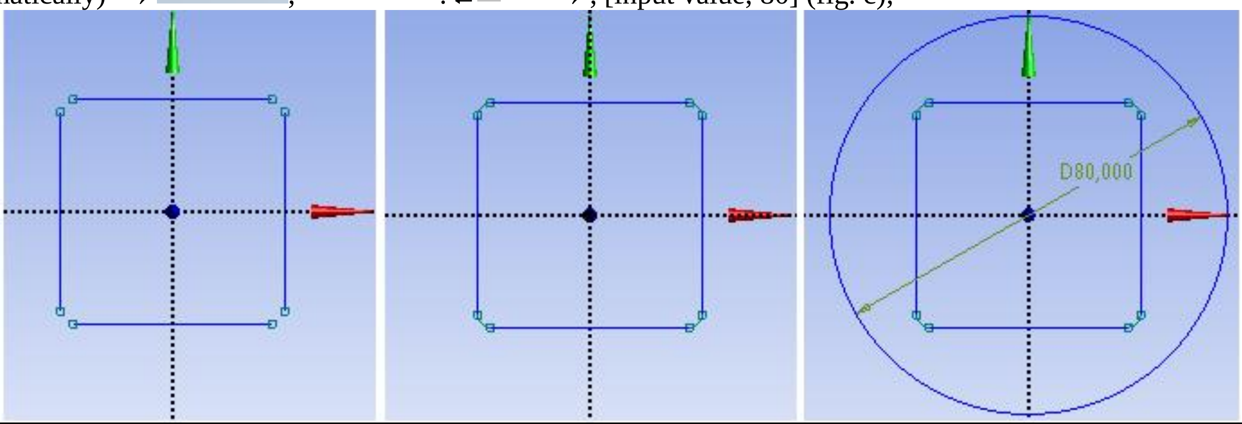
Sketching Toolboxes: ↵ Modify → ↵ Fillet → Radius: , [input value, 2,5] → [two adjacent lines are selected with ↵ at a time and the connections automatically appear] (fig. b).

Generating the circular line

Sketching Toolboxes: ↵ Draw → ↵ Circle → [select with ↵ the center of the circle coinciding with the center of the coordinate system (symbol P appears), move the cursor outwards and, as a result, release ↵ the circular line appears] (fig. c).

Dimensioning of the circle

Sketching Toolboxes: ↵ Dimensions → ↵ Diameter → [select with ↵ the circular line] (dimension appear automatically) → Details View, Dimensions: : ↵☐ D3 → , [input value, 80] (fig. c);



a.	b.	c.

C.3.4 Generate sketch 3 (arm)

Setting the Sketch 3

↵Sketching → ↵ (New Sketch)→ (se indexează automat codul schiţei / he sketch code is automatically indexed, Sketch3).

Generating overlapping circular lines over Sketch2

Sketching Toolboxes: ↵ Draw → ↵ Circle [select with ↵ the center of the circle coinciding with the center of the coordinate system (symbol P appears), move the cursor outward until it overlaps the circle in Sketch2 (change color automatically) and, as a result, release ↵ new circular line appears] (fig. a); Sketch2 → ↵ Hide Sketch (hide sketch); XYPlane → ↵ Sketch3 (activate sketch).

Generating circular lines

Sketching Toolboxes: ↵ Draw → ↵ Circle → [select with ↵ the center of the first circle coinciding with a point on the X axis (symbol C appears), move the cursor outwards and as a result of the release ↵ the circular line appears] → [select with ↵ the center of the second circle coinciding with the center of the first (the symbol P appears), move the cursor outwards and as a result of the release ↵ the circular line appears] (fig. a).

Generating straight lines (tangents)

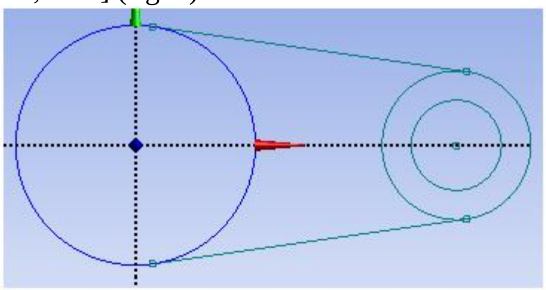
Sketching Toolboxes: ↵ Draw → ↵ Line by 2 Tangents → [the two circles are selected with ↵ in a row and the tangent lines automatically appear] (fig. b).

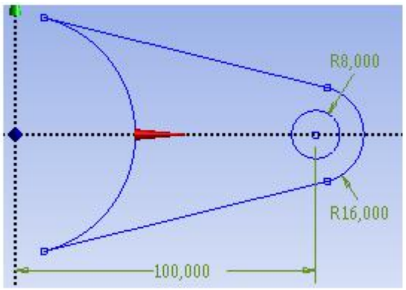
Generating contour

Sketching Toolboxes: ↵ Modify → ↵ Trim → [the parts that do not belong to the contour are deleted] (fig. a, b).

Dimensioning the new entities

Sketching Toolboxes: ↵ Dimensions → Horizontal → [select with ↵ the Y axis and the center of the crank arm bore and the dimension is automatically displayed] → Details View, Dimensions: , H → , [input the value, 100] (fig. b); ↵ Radius → [select the circle with ↵ and the dimension is automatically displayed] → Details View, Dimensions: , R → , [input the value, 8/16] (fig. b).


a.



b.

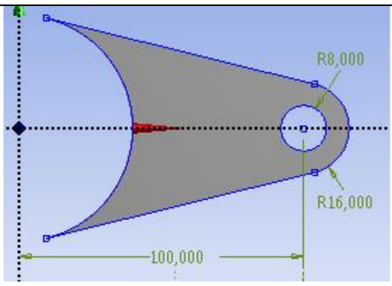
C.3.5 Generating of the surface and body of the crank arm

Generating of the surface for the arm

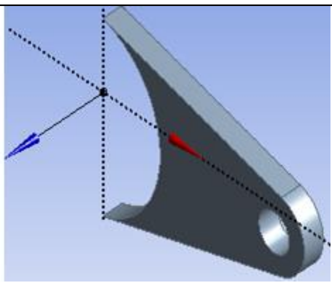
DM: ↵Concept → ↵ Surfaces From Sketches → Tree Outline: ↵ XYPlane → ↵ Sketch3 → Details View, Details of SurfaceSk: ↵ Base Objects → ↵ Apply; ↵ Generate (generate the surface, fig. a); Sketch3 → ↵ Hide Sketch (hide Sketch).

Generating of the solid body of the arm

↵ Extrude → ↵ Surface Body → Details View, Details of Extrude: ↵↵Geometry → ↵ Apply; FD1, Depth (>0) → [input the value, 10]; ↵ Generate (generate the solid, fig. b);



a.



b.

C.3.6 Generating the surface and body of the crank hub

Generating the surface for the hub

: ↵Concept → ↵ Surfaces From Sketches → Tree Outline: ↵ ⊞ XYPlane → ↵ Sketch2 → Details View, Details of SurfaceSk: ↵ Base Objects → ↵ Apply; ↵ Generate (surface generating, fig. a); ↳ Sketch2 → ↵ Hide Sketch (hide Sketch).

Generating of the solid body of the hub

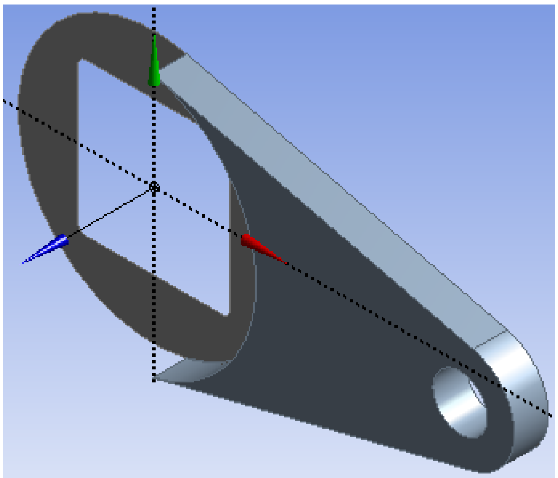
↵ Extrude → ↵ Surface Body (the surface associated with the hub)→ Details View, Details of Extrude: ↵↵ Geometry → ↵ Apply; ☐ FD1, Depth (>0) → [input the value, 30]; ↵ Generate (generate the solid, fig. b);

Obs. Because in the window that defines extrusion (Details View (Details of Extrude:) in the line Operation remained, the body of the generated hub is concatenated with the body of the previously generated arm.

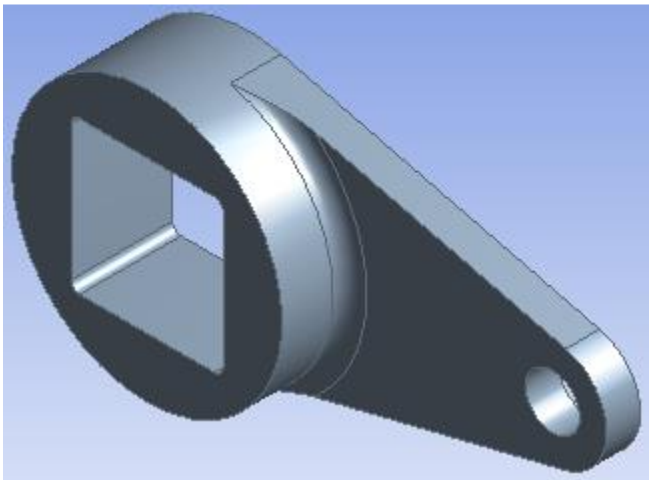
Tree Outline: ↵ Body → Details View, Details of Body: Body → [input name, Manivela].

Generating the radius connection

↵ Blend → ↵ Fixed Radius → Details View, Details of FBlend1: ↵☐ FD1, Radius (>0) → [input the value, 5] ; ↵ Manivela → ↵ (edge entity activation) → [it will be selected with ↵ the edge for connection]; ↵↵Geometry → ↵ Apply; ↵ Generate (generate radius, fig. b)



a.



b.

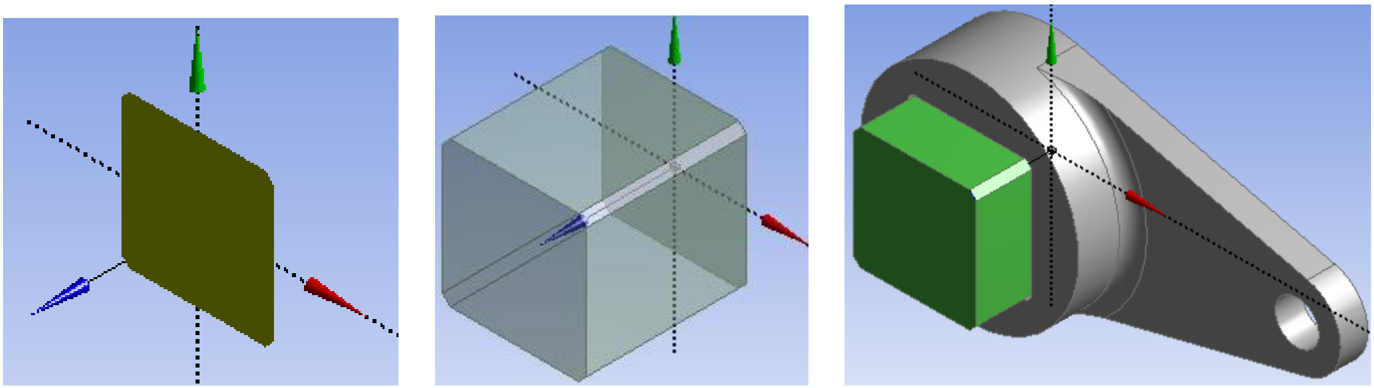
C.3.7 Generating of the surface and body of the shaft

Generating the surface for the shaft

: ↵Concept → ↵ Surfaces From Sketches → Tree Outline: ↵ ⊞ XYPlane → ↵ Sketch1 → Details View, Details of SurfaceSk: ↵ Base Objects → ↵ Apply; ↵ Generate (generate surface, fig. a); ↳ Sketch1 → ↵ Hide Sketch (hide Sketch).

Generating the solid body of the shaft

↵ Extrude → ↵ Surface Body (the surface associated with the shaft)→ Details View, Details of Extrude: ↵↵ Geometry → ↵ Apply; FD1, Depth (>0) → [input the value, 50]; ↵↵Direction Vector → ↵ XYPlane (normal plane) → ↵ Apply; ↵Operation → [select from the list ↵, ↵Add Frozen] (the resulting solid body will be independent and a separate material will be associated with it); ↵ Generate (generating the solid part, fig. b, c); ↵ Surface Body → ↵ Suppress Body (supress body).



a. b. c.

C.3.8 Save of geometric model

DM: ↵ (Save Project) → ↵ (Close).

C.4. Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics

Launching of the finite element modeling module

WB, Project Schematic: ↳ Model → ↵ Edit... → [launch modul *Mechanical [ANSYS Multiphysics*].

Setting the unit of measure system

M: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

Setting the material characteristics

Outline: ↵ Geometry → ↵ Manivela → Details of "Manivela", Material: ↵Assignment → [select from list with ↵, ↵ E335]; ↵ Arbore → Details of "Arbore", Material: ↵Assignment → [select from list with ↵, ↵ C45];

Disabling redundant entities

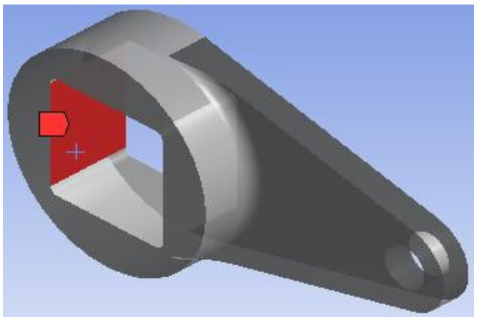
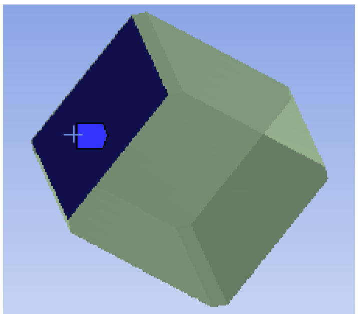
Outline: ↳ ? Surface Body → ↵ Suppress Body (deactivation of surface type entities).

Obs. In the specification tree, we observe, as a consequence of the connections between the two bodies, that a connection Contact Region has been automatically generated in the subdivision Connections, which will be further personalized.

C.4.2 Modelling the friction connections

M, Outline: ↵ Connections → ↵Insert → ↵ Manual Contact Region → Details of "Bonded - No Selection To No Selection", Definition: ↵Type → [select from list with ↵, ↵Frictional]; → ↳ Arbore → ↵ Hide Body (hide the solid Arbore) → ↵ →[with a face of the square profile of the crank body is selected, fig.a] → Details of "Frictional - No Selection To No Selection", Scope: ↵ Contact → ↵ Apply (option Contact Bodies is automatically indexed, Manivela); ↳ Arbore → ↵ Show Body → ↳ Manivela → ↵ Hide Body (hide the solid, Manivela) → ↵ → [is selected with ↵ in front of the square profile of the Arbore body, fig. b] → Details of "Frictional - Arbore To Manivela", Scope: ↵Target → ↵ Apply (option Target Bodies is automatically indexed, Arbore); ↵Definition: ↵Behavior → [select with ↵, ↵Symmetric]; Friction Coefficient → [input value, 0,2]; ↵ Advanced → ↵Formulation → [select with ↵, ↵ Augmented Lagrange] (method of solving the nonlinear model).

For each of the other 3 contacts, the sequence will be followed: ↳ Contacts → ↵Insert → ↵ Manual Contact Region → ... (continue according to the sequence above, but for another pair of surfaces:,). After browsing the above sequences in the specification tree Contacts it appears Frictional - Manivela To Arbore for four times.
Obs. For a good convergence of the solution, the entities (surfaces or edges) belonging to the fixed bodies, to the bodies with increased material rigidity (greater longitudinal elasticity modulus) or smaller curves are adopted in the window Details of "Frictional at the option.

a. ***b.***

C.4.3 *Set discretization parameters, model discretization and analysis type setting*

<u>Setting the local discretization parameters in the contact areas</u>

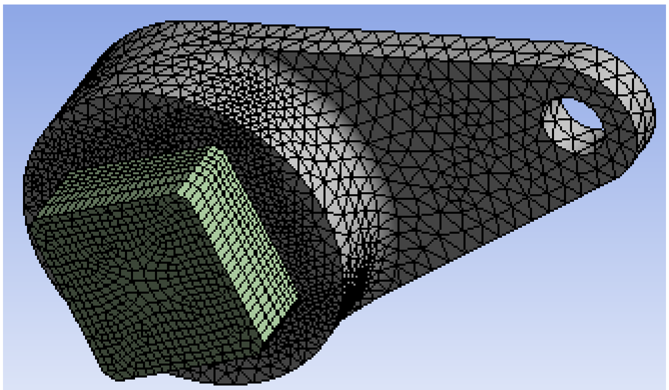
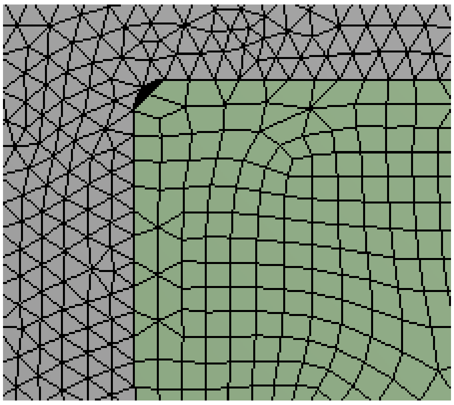
, Outline: ↵ Mesh → Details of "Mesh", ↵ **Defaults**: ↵ Relevance → [will be adopted with ↵ the value, 100]; ↵ **Sizing**: ↵Relevance Center → [select from list ↵, ↵Fine].

<u>Setting the local parameters in the contact areas</u>

↳ Mesh → ↵ Insert → ↵ Contact Sizing → Details of "Contact Sizing" - Contact Sizing, **Scope**: ↵Contact Region → [select from list ↵, ↵Frictional - Arbore To Manivela]; **Definition**: ↵ Element Size → [input the value, 2] (this sequence is repeated for the other three contacts).

<u>Automatic discretization</u>

↳ Mesh → ↵ Generate Mesh (fig. a, b).

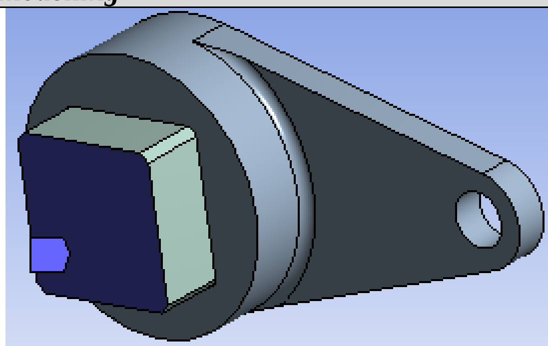



a. ***b.***

C.5. *Supports and restraints modelling*

<u>Generating of the constraint type (cancels all 6 degrees of mobility)</u>

, Outline: ↵ **Static Structural (A5)** → ↵ Supports → ↵ Fixed Support; ↵ **Model (A4)** → ↵ → [select with ↵+Ctrl faces with constraint (frontal face of Arbore)]; ↵ Fixed Support → Details of "Fixed Support", **Scope**: ↵ Geometry → ↵ No Selection → ↵ Apply.



C.6. *Loads modelling*

Generating of force load on the surface

, Outline: ↳ Static Structural (A5) → ↵ Insert → ↵ Force →

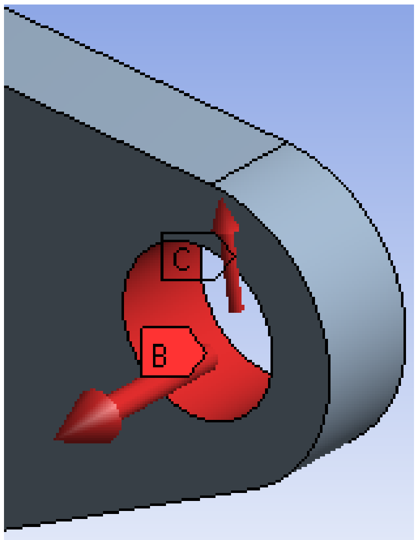
Details of "Force", Scope: ↵Geometry → ↵ → [the cylindrical surface of the bore in the crank arm shall be selected with ↵] → ↵ Apply;

Definition: ↵Define By → [select from list with ↵, ↵Components]; ↵ Z Component → [input the value, 400] (force B).

Generating of load with force in the bearing

↳ Static Structural (A5) → ↵ Insert → ↵ Bearing Load →

Details of "Bearing Load", Scope: ↵↵Geometry → ↵ → [the cylindrical surface of the bore in the crank arm shall be selected with ↵] → ↵ Apply; Definition: ↵Define By → [select from list ↵, ↵ Components]; ↵ Y Component → [input the value, 15000] (force C).



D. SOLVING THE FEA MODEL

D.1 Setting the convergence criterion for solving the nonlinear physical model (with friction)

, Outline: → ↵ Solution (A6) → ↵ Solution Information, Details of "Solution Information",

↵ Solution Information: ↵Solution Output →[select from list with ↵, ↵Force Convergence] (the criterion of force convergence is adopted).

D.2 Setting the results

Setting the total displacement

, Outline: ↳ Solution (A6) → ↵ Insert → ↵Deformation → ↵ Total.

Setting the equivalent stress

↳ Solution (A6) → ↵ Insert → ↵Stress → ↵ Equivalent (von-Mises).

Setting the structural error

↳ Solution (A6) → ↵ Insert → ↵Stress → ↵ Stress → ↵ Error.

Setting parameters from contacts

↳ Solution (A6) → ↵ Insert → ↵ Contact Tool → ↵ Contact Tool;

↳ Contact Tool → ↵Insert → ↵Frictional Stress / Pressure / Sliding Distance / Penetration / Gap / Status.

Setting the design parameters

↵ Solution (A6) → ↵ Tools ↵ → ↵ Stress Tool;

↳ Stress Tool ↵Insert → ↵Stress Tool → ↵ Safety Factor / Safety Margin / Stress Ratio.

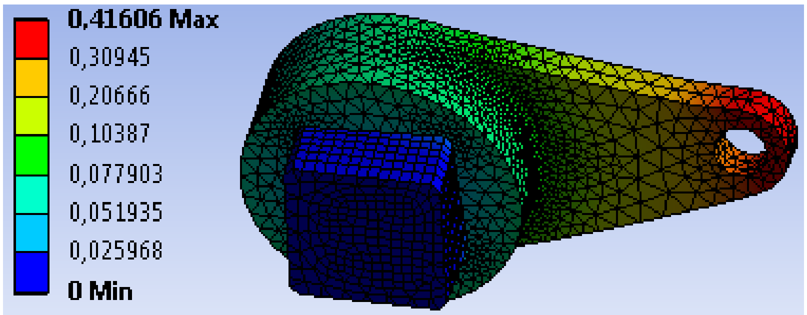
D.3 Launching the solving module

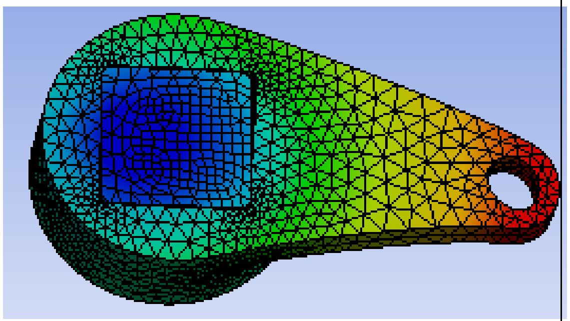
, Outline: ↵ Solution (A6) → ↵ Solve.

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

, Outline: ↵⊞···? Solution (A6) → ↵···✓ Total Deformation (fig. a); ↵ ISO (axonometric visualization); ↵ → [select from list with ↵ ▼, ↵ Smooth Contours] (visualization of smooth contours); ↵ → [select from list with ↵ ▼, ↵ Show Elements] (visualisation the FE structure). ↵ Result → [select from list with ↵ ▼, ↵ 44 (2x Auto)] (select the scale factor); Graph → ↵ Animation ▶ ■ (view the animation).

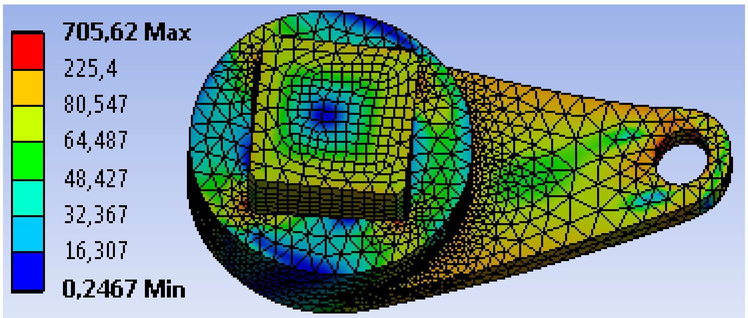




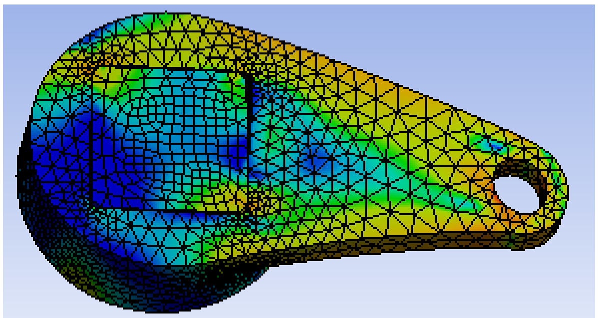
E.2 Viewing the equivalent stress field

Viewing the equivalent stress field

, Outline: ↵⊞···? Solution (A6) → ↵···✓ Equivalent Stress .



a.



b.

Viewing the equivalent stress field in section

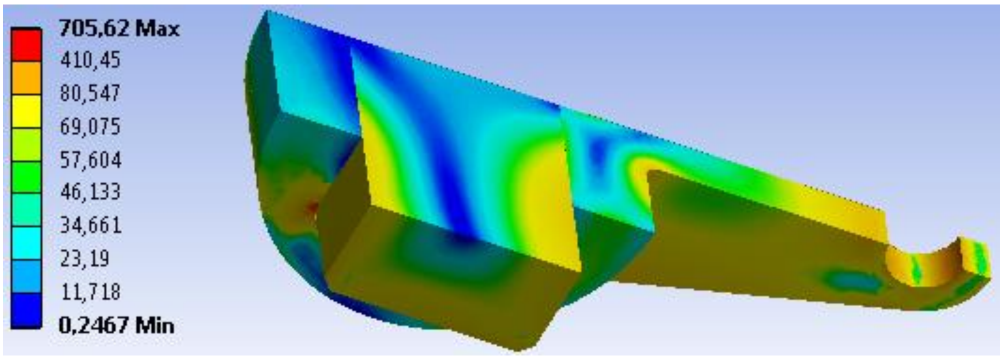
Generating of section plan

Outline: ↵⊞···✓ Coordinate Systems → ···✓ Global Coordinate System → ↵ Insert → ↵ Coordinate System → Details of "Coordinate System", **Origin** : ↵ Define By → [select from the list ↵ ▼, ↵ Global Coordinates]; ↵ Origin X/ Origin Y/Origin Z → [input the value, 0,0 (by default, usually)] **Principal Axis** : ↵ Axis → [select from the list ↵ ▼, ↵ X (implicit)]; ↵ Define By → [select from the list ↵ ▼, ↵ Global X Axis]; **Orientation About Principal Axis** : ↵ Axis → [select from the list ↵ ▼, ↵ Y (implicit)]; ↵ Define By → [select from the list ↵ ▼, ↵ Global Z Axis] .

Visualizating of the equivalent stress in the section according to the generated plane

···✓ Coordinate System → ↵ Create Section Plane ; ↵ (from main menu) → Section Planes: [activate ☑ Slice Plane 1] (automatically appear). ↵ → [select from the list ↵ ▼, ↵ Smooth Contours] (smooth contour visualisation,);

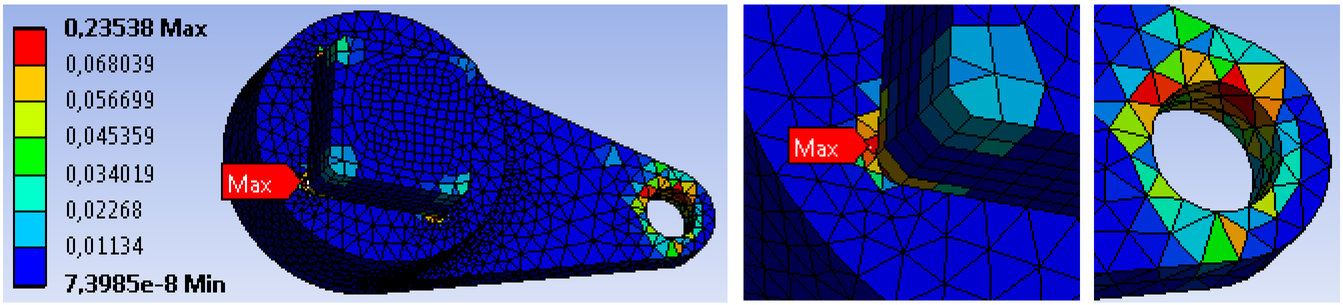
Obs. In order to delete the section plan, the succession is carried out: ↲ → Section Planes: ↲ Slice Plane 1 → ↲ ✕ → ↲.



c.

E.3 Viewing the structural error

, Outline: ↲ ? Solution (A6) ↲ Structural Error.



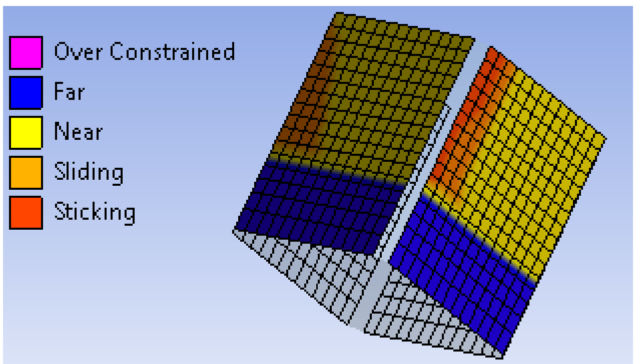
E.4 Viewing the fields of the contact parameters (state, friction stress, pressure, sliding, jump, penetration)

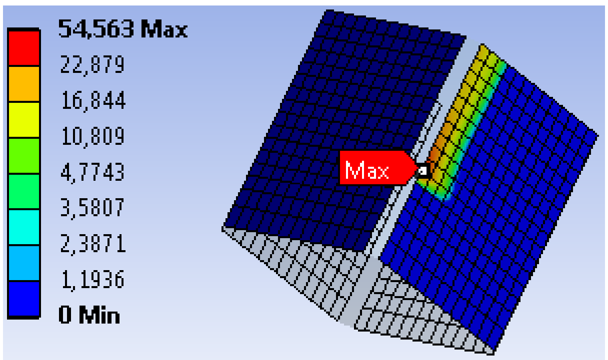
↲ Solution (A6) → ↲ Status (fig. a);

↲ Frictional Stress (fig. b); ↲ Pressure (fig. c);

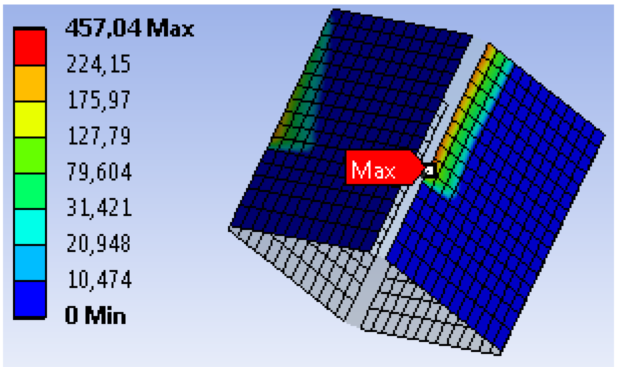
↲ Sliding Distance (fig. d);

↲ Gap (fig. e); ↲ Penetration (fig. f).

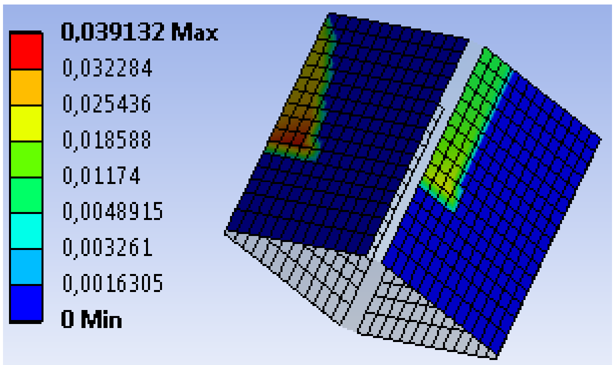




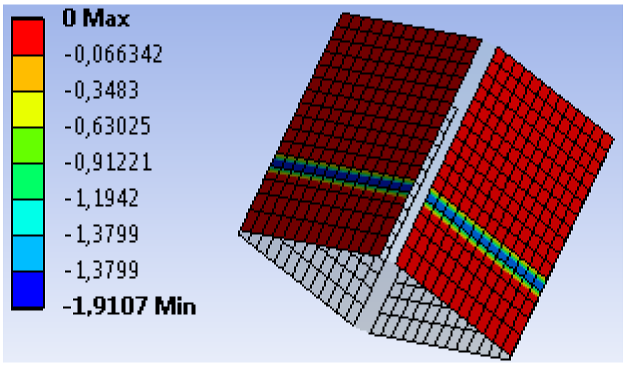
a. *b.*



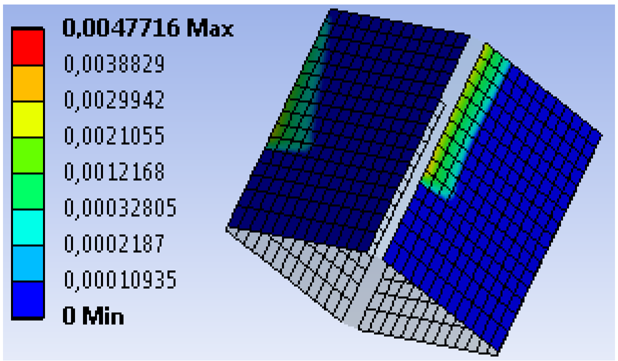
c.



d.



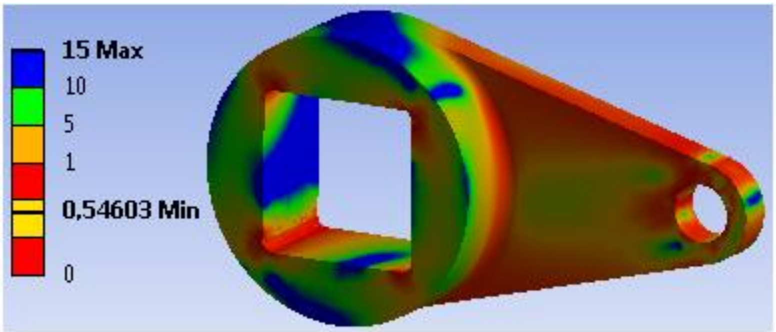
e.

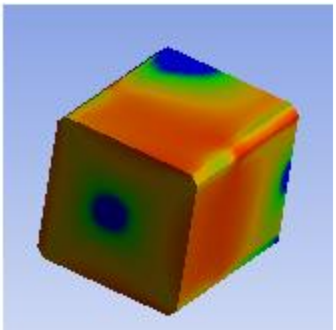


f.

E.5 Viewing the safety factor field

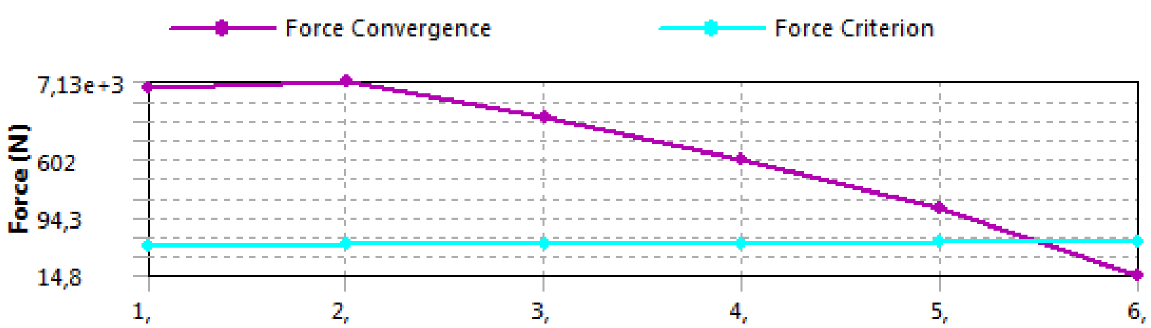
Solution (A6) → Safety Factor;





E.6 Viewing the convergence graph of the solution of the nonlinear problem

Solution Information



F. ANALYSIS OF RESULTS

F.1 Interpretation of results

Analyzing the results obtained from modeling and post-processing the results (subchapter E), the following are highlighted:

- As a result of the deformation process of the semi-finished product as a result of the action of the connecting rod (subchapter fig.), Increased displacements (max. 0.4106 mm, subchapter E.1) are observed in the extreme area of the crank.
- The equivalent stress has increased values (max 705.62 MPa; subchapter E.2, fig. A, b, c) in the areas of the connecting rod, in the corner of the square profile of the crank and in the area of connection of the arm to the hub.
- In the subcap. E.4 the states of the contacts are visualized (subchapter E.5, fig. A) and the values of some contact parameters: surface tension tangential friction - max. 54,563 MPa, fig. b; pressure - max 457.04 MPa, fig. c; relative slip - max 0.039 mm, fig. d; the game (jump) - max 1.91 mm, fig. e; penetration - max 0.00477 mm, fig. f. These values are useful for designing the shaft-hub assembly (eg starting from the maximum pressure value determines the hardness of the active contact surfaces).
- Highlighting the field of the safety factor related to the allowable values (subchapter) is particularly useful in designing to identify the areas with values of this unacceptable factor; thus, dimensional and shape corrections can be made to obtain the optimal structure (subchapter F.2).

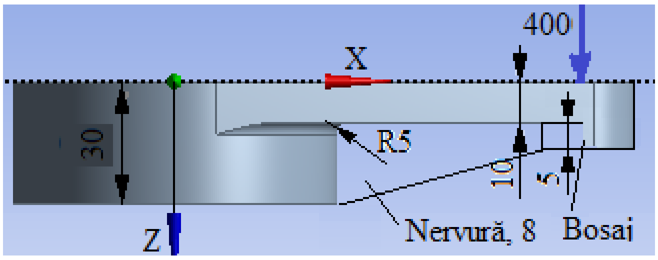
F.2 Analysis of the precision and convergence of solving the nonlinear model

Following the analysis of the obtained results, related to precision and convergence, as a result, of the modeling and FEA (subcap. E.3 and E.6) the following are highlighted:

- The maximum structural error in the connection areas of the square profile of the connecting rod has a reduced value (0.23538 mJ corresponding to the areas with increased values of the equivalent stress) shows an accuracy of the acceptable results (stresss).
- The convergence of the model solution is done quickly in 6 steps (subchapter E.6) and the calculation time is reduced.

F.3 Design studies

In order to avoid the occurrence of the rupture micro-cracks in the connection areas of the square profile, a fact highlighted by increased values of the equivalent stress (subchapter E.2) but also by subunit values of the safety factor (subchapter E.6) are imposed, on on the one hand, *dimensional changes* (eg, increase of the radius of connection of the square profile; increase of the radius of the bore in the arm; increase of the radius of connection of the arm to the hub and / or increase of the thickness of the arm or, on the other hand, *changes of the shape*.



In case of dimensional changes, it is necessary to modify the analysis model and to solve the model by going through the successions: DM, Tree Outline: …… change the dimension value↵ Generate ; M, Outline: ↳ ⊞ Geometry → ↵ Refresh Geometry; ↵ Solve . After the model is solved, the results are re-analyzed and reinterpreted. If after the stage of dimensional changes the crank structure is not optimal (minimum weight and equivalent stresses, respectively, safety factors in permissible fields), the shape of the model is changed (eg adopting an assembly after a hexagonal profile and / or modification of the arm shape by the insertion of a bump and / or a rib).

G. CONCLUSIONS

The modeling and analysis with finite elements of this paper were also done with didactic purpose following the initiation of the user with the main stages of development of an application of FEA in ANSYS Workbench, which insists, in particular, on the modeling and analysis of a deformable element and of its contacts with another adjacent element.

The adopted FEA model involves considering the multiple surface-to-surface friction contacts of a square / hexagonal joint between two elements of linear behavior. The loads are introduced distributed on a cylindrical surface and considering the existence of the bearing.

As a result of solving the model with finite elements nonlinearly adopting the method of force convergence, results have been obtained with increased precision, the values of the obtained parameters (displacements, stresses, safety factors) being useful for designing the crank element considering the profiled assembly with square profile.

Application: FEA-A.8
Threaded assembly

KEY WORDS
Linear static analysis, Axial symmetrical state of tension, Linear material, 2D geometric model, 2D finite element, Linear finite element, Mechanical friction contact, Structural error, Threaded assembly, Mechanical subassembly

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introducere / Introduction

FEA, as a general method of studying the physical phenomena and processes in mechanical structures, also allows the analysis of mechanical fields that occur in the case of *mechanical assembly contacts* that involve consideration of elastically deformable surfaces in direct contact and sliding friction between them.

The *threaded connections* frequently used in the construction of removable screw-nut assemblies form complex spatial structures involving mechanical contacts with friction and severe stress concentrations, difficult to determine with classical theoretical and / or experimental methods, can be analyzed more accurately by modeling and FEA .

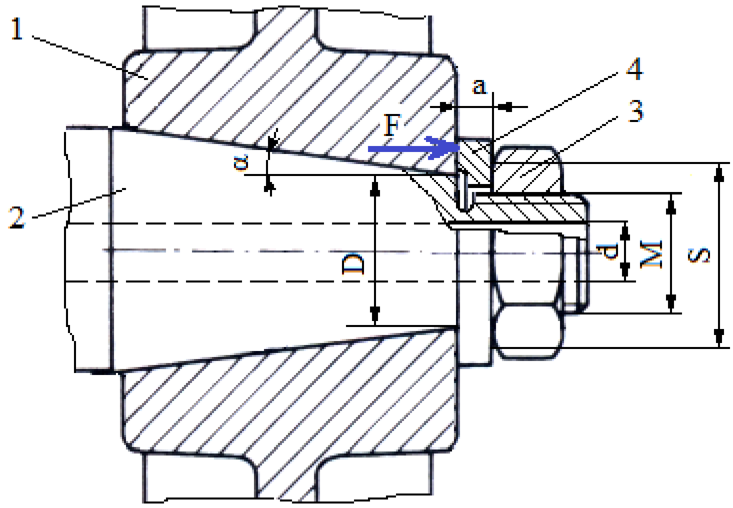
A.2 Application description

In order to achieve the necessary tightening of the shaft-hub assembly on the conical surface (fig. a) it is necessary to develop a pressing force F by tightening the nut 4 with internal thread (fig. b) in relation to the external thread practiced on the shaft 2.

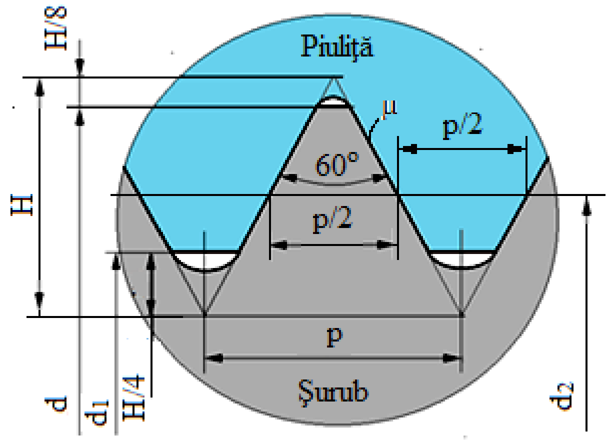
The metric fixing threads have the profile angle 60° and the theoretical height H = 0.866 p, where p is the thread pitch. The contact surfaces are delimited by cylindrical surfaces with diameter d1 on the inside and diameter d_2 on the outside, respectively.

In addition, the threaded assembly is described by the medium (virtual) cylinder with diameter d2 on which the thickness of the nut thread turn is equal to the thickness of the screw thread turn (p / 2).

For functional and technological reasons, the helical surfaces are connected inside (nut) and outside (screw). The transmission of force from the nut to the screw by shape (direct contact) to screwing between the elastically deformable helical surfaces involves relative micromovements with friction.



a.



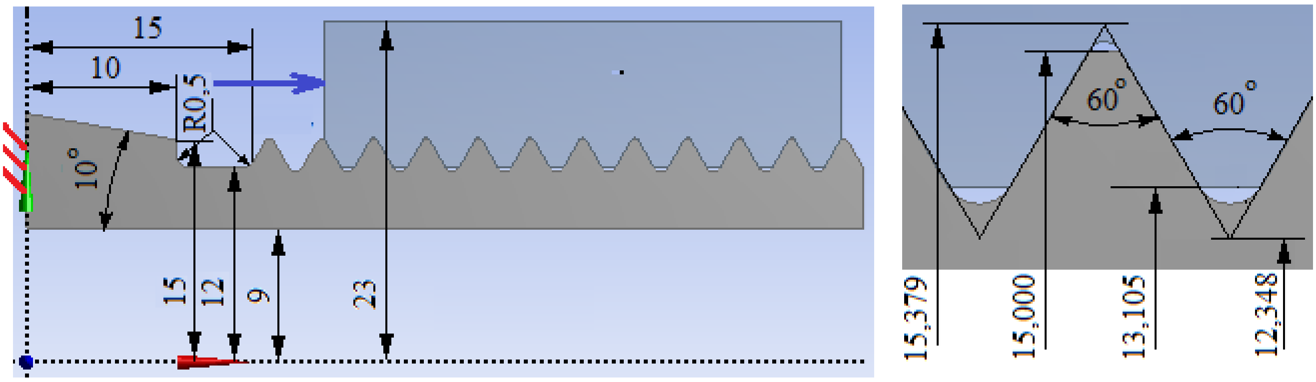
b.

A.3 The application goal
In this application, the FEA of the displacement and tension fields in the area of the threaded assembly with M = 30 mm and the pitch p = 3.5 mm is required. For the area adjacent to the threaded assembly are considered: S = 46 mm, d = 18 mm, a = 10.25 mm, D = 30 mm, α = 10^0. The assembly is loaded with axial force F = 25000N. The shaft 1 and the nut 3 are made of heat-treated construction steel (E235).

B. THE FEA MODEL

B.1 The model definition
In order to design the FEA model of the nut / screw in interaction, it is necessary to consider two adjacent areas of the two elements adopting the following simplifying hypotheses: • considering that there are no significant variations on the circumference of the physical parameters (displacements and stresses), a planar model can be adopted that can be framed in the axial-symmetrical state of stresses. • existing friction in mechanical contacts, • adoption of material strength constraints (embedding, action of force distributed on the surface), • the material has an elastic linear behavior, • the deformation takes place statically (the variation of the deformation force over time is not taken into account).

B.2 The analysis model description
In order to simulate the behavior of the threaded assembly, it consider the axial section with the dimensions in the figure below. The geometric modeling of the thread is based on the approximate pattern in the subcap. A.2, fig. b, where for H = 0.866p = 0.855 * 3.5 = 3.031 mm, d_1 = 26.211 mm is obtained. The fillets of the nut and screw profiles are obtained by automatic generation considering that the connection spring is tangent to the profile lines. The thread will be generated by multiplying in the axial direction (12 turns for the screw and 10 turns for the nut). For this analysis, the structure is axial-symmetrical and it is modeled with 2D finite elements. In order to simulate the behavior of the assembly as close as possible to reality, the friction between the assembled elements will be taken into account, the coefficient of friction μ = 0.2. The load will be made on the front surface of the nut with F = 15000 N.



a. *b.*

B.3 Characteristics of the material and the environment
For linear static analysis the following resistance characteristics of E335 material are considered: • longitudinal modulus of elasticity, E = 206000 N / mm^2; • Poisson's ratio, ν = 0,3. Average working temperature of the subassembly, T_0 = 20 ° C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project
<u>*Creating of the project*</u> WB, Toolbox: ↵⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural in *FEA-A.8*]. <u>*Problem type setting (2D)*</u> A: ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from the list ↵▼, ↵ 2D] → [close the window ↵ X]. <u>*Save of the project*</u> ↵ Save As... → WB Save As, File name: [input name, FEA-A.8] → ↵ Save.
C.2 Modelling of material and environment characteristics
WB, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, Young's Modulus, [select from column C (Unit) cu / with ↵▼, ↵MPa], [input in column B (Unit) valoarea / value, 206000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).
C.3 Geometric modelling
C.3.1 Model loading, DesignModeler (DM)
WB, Project Schematic: ↳ Geometry → ↵ DM New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.
C3.2 Sketch generation, screw
<u>*Viewing default plane (XY)*</u> DM, Tree Outline: ↵Sketching → (Look at face/Plane/Schetch) [automatically view of default plane XY Plane]; <u>*Generating of horizontal and vertical lines*</u> ↵Draw → ↵ Line → [horizontal and vertical lines are generated by activating with ↵ the end points of each line respecting the conditions of coincidence with the horizontal direction (symbol H appears automatically), respectively vertical (symbol V appears automatically)] (fig. a). <u>*Cuting lines at the edge*</u>

↵Sketching → ↵ Trim → [select with ↵ the parts of the end of the lines separated by the vertical line] (fig. b).

Partial sketch dimensioning

Dimensioning in the horizontal direction

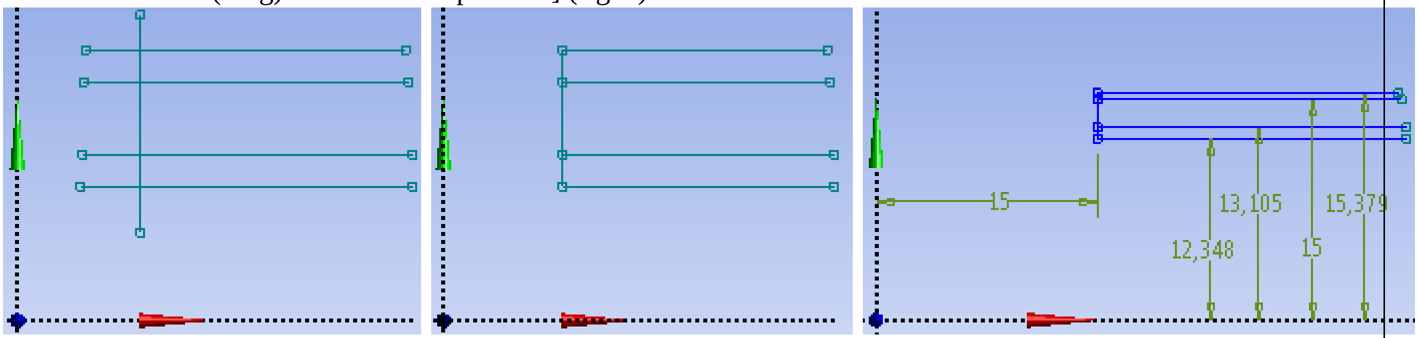
Sketching Toolboxes: ↵Dimensions → ↵ Horizontal → [select with ↵ the line parallel to the Y axis and the Y axis] (the dimension is automatically displayed) → Details View, **Dimensions:**, ☐ H → [input value, 15] (fig. c);

Dimensioning in the vertical direction

↵ Vertical → [select with ↵ the line parallel to the X axis and the X axis] (the dimension is automatically displayed) → Details View, **Dimensions: 2**: ↵☐ V → [input value, 12,34775/15/13,1055/15,378875] (the sequence is repeated for the other lines, fig. c);

Edit dimensions

↵ Display → ↵Name: ☑ (dezactivate) → ↵ Name: ☐ Value: ☑ (activate, fig. c); ↵ Move → [select the dimension with ↵ and move (drag) to the desired position] (fig. c).



a. **b.** **c.**

Reference thread contour generation (one pitch length)

Polyline generation

Sketching Toolboxes: ↵Draw → ↵ Polyline → [draw the polyline by marking the 4 points with ↵ respecting the coincidence restrictions P (first point) and C (second and third point)] → [select with ↳ a point in the graphics area] (context menu appears)→ ↵Open End.

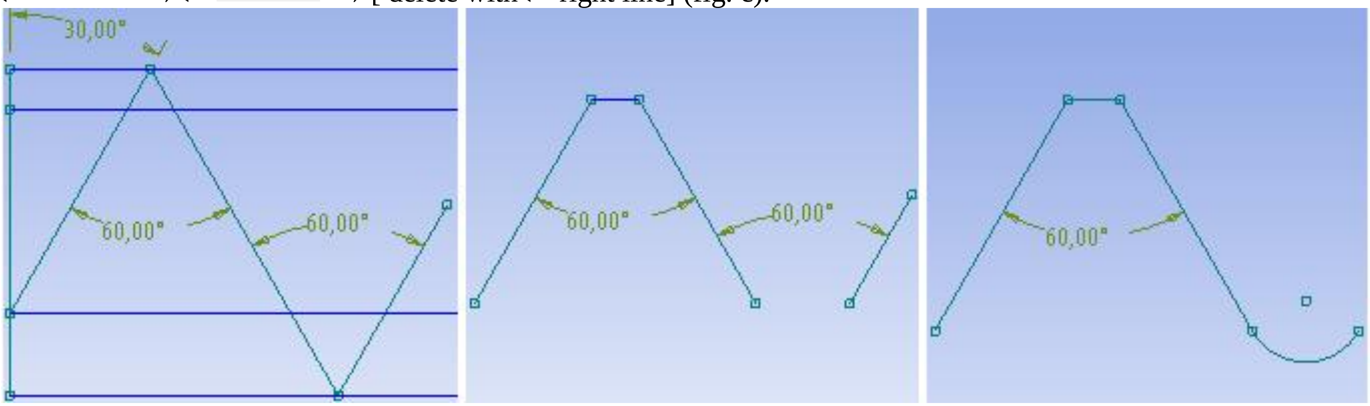
Generating profile lines

↵Sketching → ↵ Trim → [the parts that do not belong to the outline are deleted with ↵] (fig. a, b).

↵Draw → ↵ Arc by 3 Points → [select with ↵ the two marginal points of the arc respecting the coincidence constraint, the symbol P, and the third point will be marked in the opposite area the center of the arc (towards the intersection point of the straight lines) after the tangent restrictions to the straight lines appear (twice the symbol T)] (fig. b, c).

Final contour generation

↵Sketching → ↵ Trim → [delete with ↵ right line] (fig. c).



a. **b.** **c.**

Generation of the screw thread by multiplying the reference contour

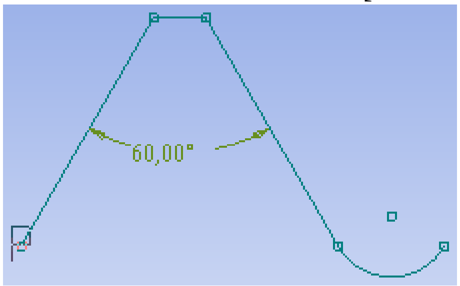
Multiply the reference contour

↵Modify → ↵ Replicate → [value, 0] (appear: r = 0 ° f = 2) → [will be selected with ↳ a point in the graphics area] (context menu appears) → ↳Selection Filter → ↵ 2D Edge → [will be selected with

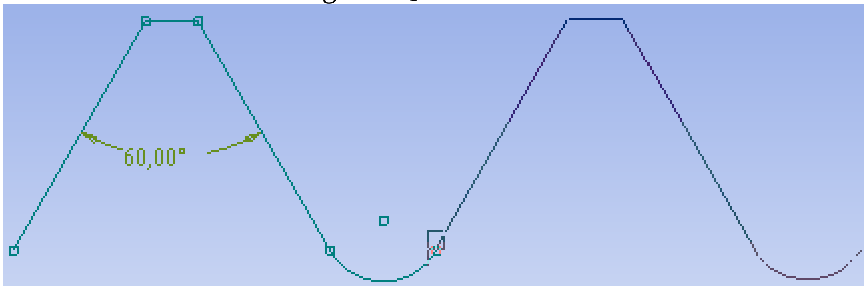
↳ a point in the graphics area] (context menu appears) → ↵ End / Set Paste Handle → ↳ Selection Filter → ↵ Point → [select with ↵ the point on the left respecting the coincidence constraint (symbol P appears, fig. a)] (the set of lines multiplies in the graphics area) → [move the set of lines and mark with ↵ the point on the right respecting the restriction coincidence (P symbol appears, fig. b)] (the multiplied set appears, this sequence is repeated 11 times, fig. c) ↵ context menu) → ↵ End (fig. c).

Generating the final thread contour

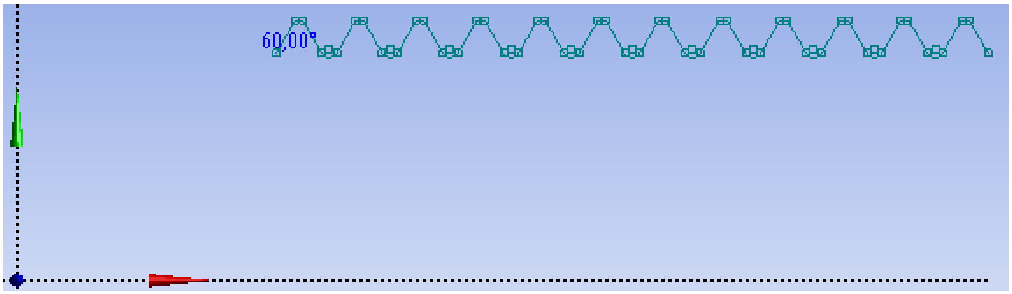
↵ Modify → ↵ Trim → [is deleted with ↵ the last connecting curve].



a.



b.



c.

<u>Screw pattern contour generation and dimensioning</u>

Contour generation with polyline

↵ Draw → ↵ Polyline → [draw the polyline marking the points with ↵ respecting the coincidence constraints P, H, V and C]; at the end, after selecting the last point, activate the context menu with ↳ and select the option ↵ Open End] (fig. a).

Dimensioning in the horizontal direction

Sketching Toolboxes: ↵ Dimensions → ↵ Horizontal → [select with ↵ the line parallel to the Y axis and the Y axis] (the dimension is automatically displayed) → Details View, **Dimensions:** , ☐ H → [input value, 10] (fig. a);

Dimensioning in the vertical direction

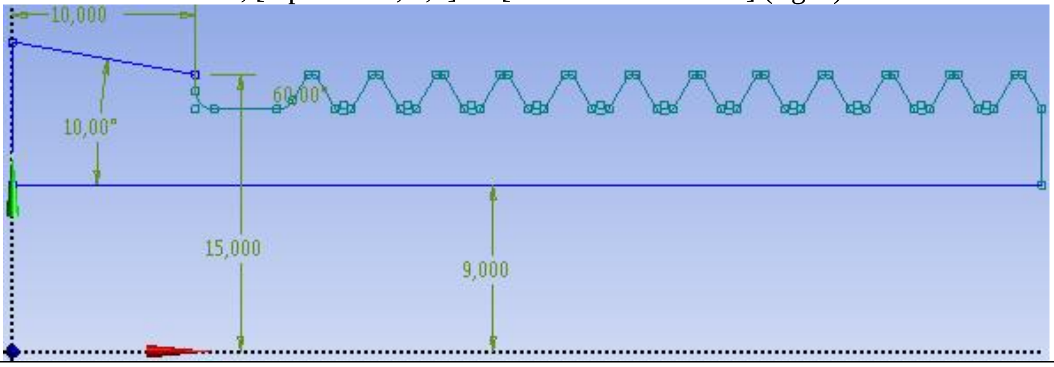
↵ Vertical → [select with ↵ the line parallel to the X axis and the X axis] (the dimension is automatically displayed) → Details View, **Dimensions:**: ↵ ☐ V → [input value, 9/15] (the sequence is repeated for the other lines, fig. a).

Dimensioning of the angles

↵ Angle → [is selected with ↵ angle lines] (the dimension is automatically displayed) → Details View, **Dimensions:**: ↵ ☐ A → [input value, 10] (fig. a).

Generate the fillets

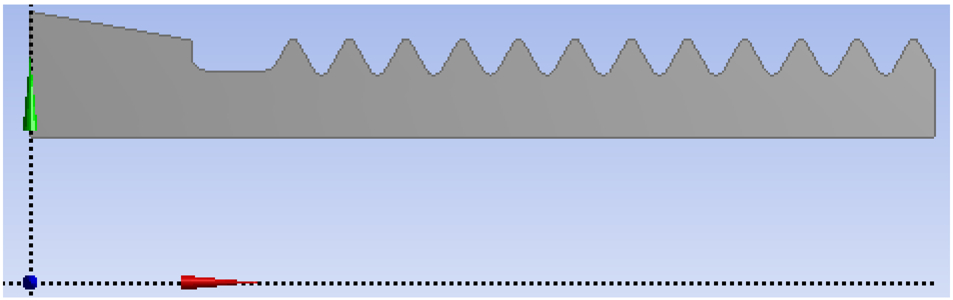
↵ Modify → ↵ Fillet → Radius: , [input value, 0,5] → [select the fillet lines] (fig. a).



a.

C.3.3 Generating of the screw surface

DM: ↵ Concept → ↵ Surfaces From Sketches → ↵ Sketch1 → Details View, Details of Surface: ↵ Base Objects → ↵ Apply; ↵ Generate (generate surface, fig. a); ↳ Sketch1 → ↵ Hide Sketch (sketch masking). ↵ Surface Body → Details View, Details of Surface Body: ↵ Body, [input name, Şurub].



a.

C.3.4 Nut sketch generation

Nut sketch initialization

DM: ↵ (New Sketch) → (the object is automatically indexed in the specification tree Sketch2).

Generating reference thread contour nut

Activate screw sketch

Tree Outline: ↵ Sketch1 → ↵ → ↵ (Display Model).

Generate preliminary contour

↵ Sketching → Sketching Toolboxes: ↵ Draw → ↵ Polyline → [the polyline will be drawn by selecting with ↵ the points of the screw thread respecting the coincidence conditions P] (fig. a).

Screw sketch masking

↵ Modeling → Tree Outline: ↳ Sketch1 → ↵ Hide Sketch → ↵ (Display Model).

Delete line

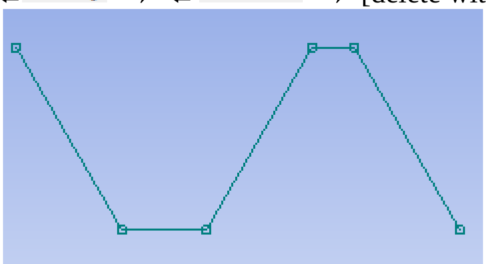
↵ Modify → ↵ Trim → [delete with ↵ the last one in the thread connection area] (fig. b).

Generating thread connection arc

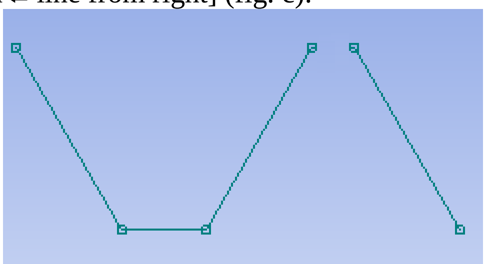
↵ Draw → ↵ Arc by 3 Points → [select with ↵ the two marginal points of the arc respecting the coincidence constraint, the symbol P, and the third point will be marked in the opposite area the center of the arc (towards the intersection point of the straight lines) after the tangent restrictions to the straight lines appear (twice the symbol T)] (fig. b, c).

Delete line

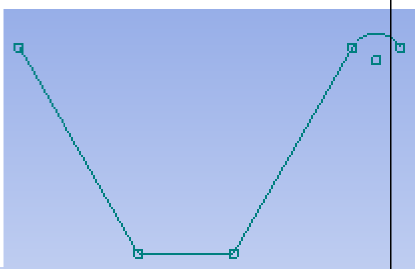
↵ Modify → ↵ Trim → [delete with ↵ line from right] (fig. c).



a.



b.



c.

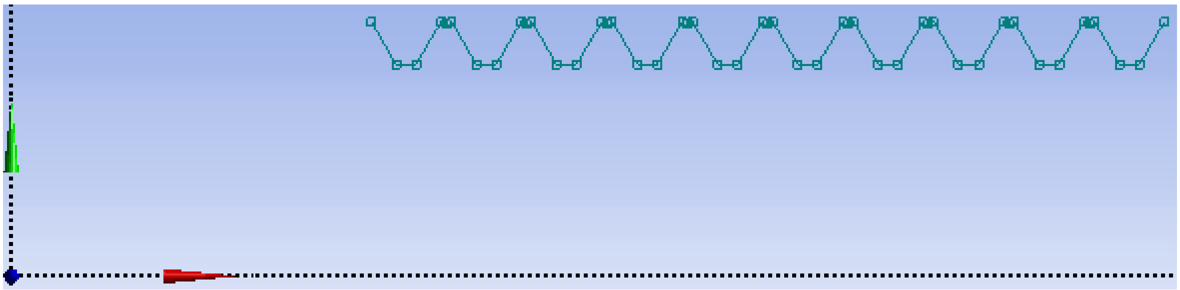
Generating nut thread by multiplying reference contour

Reference contour multiplication

↵ Modify → ↵ Replicate → [input in r = valoarea, 0] (apare / appear: r = 0 ° f = 2) → [will be selected with ↳ a point in the graphics area] (context menu appears)→ ↳ Selection Filter → ↵ 2D Edge → [will be selected with ↳ a point in the graphics area] (context menu appears)→ ↵ End / Set Paste Handle → ↳ Selection Filter → ↵ Point → [select with ↵ the point on the left respecting the coincidence constraint (symbol P appears)] (the set of lines multiplies in the graphics area) ↵ [move the set of lines and mark with ↳ the point on the right respecting the coincidence constraint (appears symbol P)] (the multiplied set appears, this sequence is repeated 9 times, fig. a) → [will be selected (after the last multiplication) with any point in the graphics area] (the context menu appears)→ ↵ End (fig. c).

Delete the line

↵ Modify → ↵ Trim → [is deleted with ↵ the last connection line].



a.

Generating and dimensioning of the nut pattern contour

Contour generation

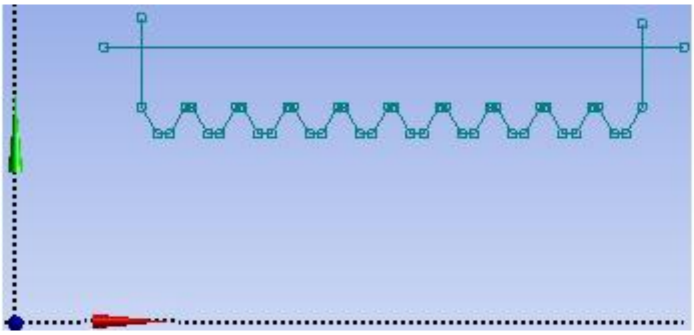
↵ Draw → ↵ Line → [draw 2 vertical lines and one horizontal line with ↵ respecting the conditions of vertical and horizontal directions, respectively symbols V and H, respectively].

Delete lines

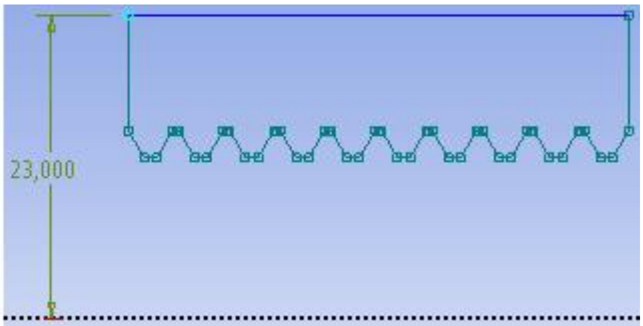
↵ Modify → ↵ Trim → [delete with ↵ the ending line].

Dimensioning on vertical direction

↵ Vertical → [select with ↵ the line parallel to the X axis and the X axis (fig. b)] (the dimension is automatically displayed)→ Details View, **Dimensions:**: ↵ V → [input value, 23] (fig. b).



a.

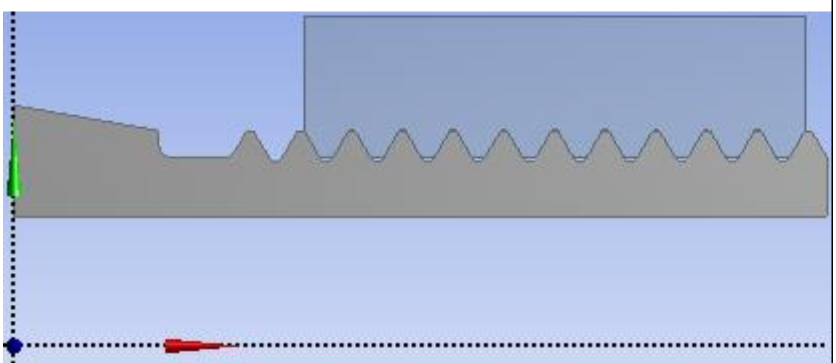


b.

C.3.5 Generating the nut surface

DM: ↵ Concept → ↵ Surfaces From Sketches → ↵ Sketch2 → Details View, **Details of Surface**: ↵ Base Objects → ↵ Apply; ↵ Generate (generate surface, fig. a); ↳ Sketch1 → ↵ Hide Sketch (sketch masking).

↵ Surface Body → Details View, **Details of Surface Body**: ↵ Body, [input name, Piuliță].



a.

C.3.6 Saving of geometric model

: ↵ (Save Project) → ↵ (Close).

C.4 Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

Launching of the finite element modeling module

, Project Schematic: Model → ↵ Edit... → [launch modul *Mechanical [ANSYS Multiphysics*].

Setting the unit of measure system

: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

Setting the material characteristics

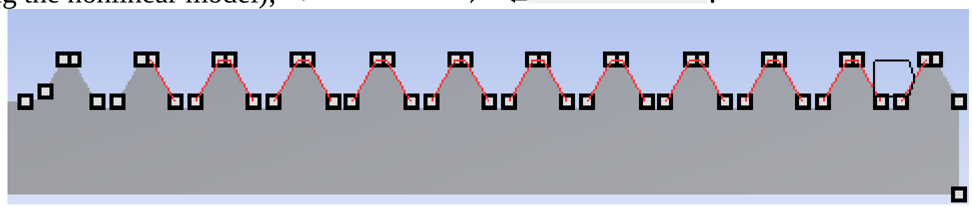
Outline: ↵ Geometry → ↵ Surub → Details of "Surub", Material: ↵ Assignment → [select from the list ↵, ↵ Structural Steel (default)]; ↵ Piulita → Details of "Piulita", Material: ↵ Assignment → [select from the list ↵, ↵ Structural Steel (default)];

Setting the model type (axial asymmetric)

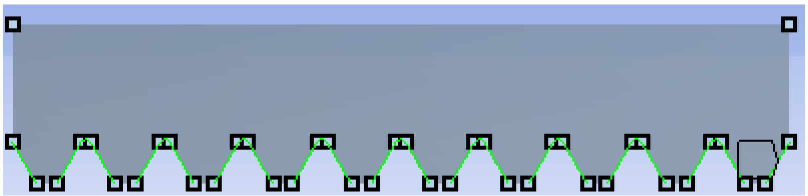
Outline: ↵ Geometry → Details of "Geometry", Definition: ↵ 2D Behavior, [select from the list ↵, ↵ Axisymmetric].

C.4.2 Modelling the friction connections

, Outline: ↵ Connections → ↵ Insert → ↵ Manual Contact Region → Details of "Bonded - No Selection To No Selection", Definition: ↵ Type → [select from the list ↵, ↵ Frictional]; Piulita → ↵ Hide Body → ↵ →[select with ↵ the thread lines of the screw threads, fig. a] → Details of "Frictional - No Selection To No Selection", Scope: ↵ Contact → ↵ Apply (option Contact Bodies will index automatically, Surub); Piulita → ↵ Show Body → Surub → ↵ Hide Body → ↵ → [select with ↵ the thread lines of the nut threads, fig. b] → Details of "Frictional - Surub To Piulita", Scope: ↵ Target → ↵ Apply (option Target Bodies will index automatically, Piulita); ↵ Definition: ↵ Behavior → [select with ↵, ↵ Symmetric]; Friction Coefficient → [input value, 0,2]; ↵ Advanced → ↵ Formulation → [select with ↵, ↵ Augmented Lagrange] (the method of solving the nonlinear model); Surub → ↵ Show Body.



a.

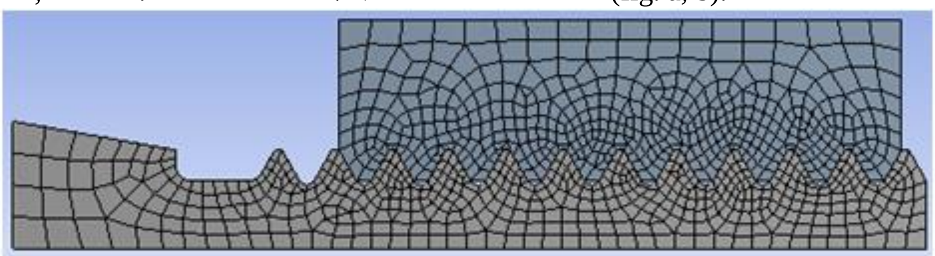


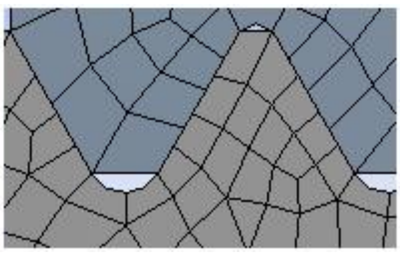
b.

C.4.3 Model discretization and analysis type setting

Automatic meshing

, Outline: Mesh → ↵ Generate Mesh (fig. a, b).





a. ***b.***

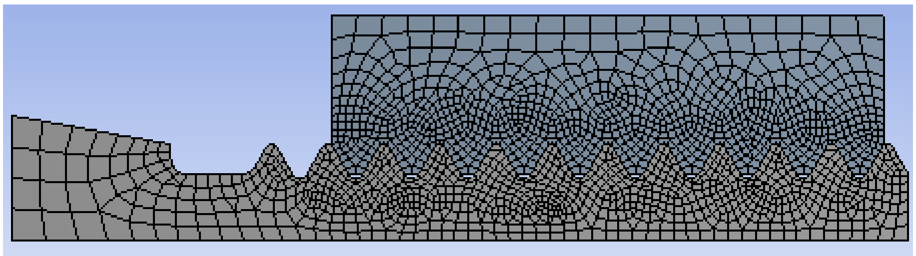
Obs. In fig. a, b there are discontinuities of the finite element structure at the level of the contact surface, which leads to an inadequate modeling of the contact phenomena and it is necessary to correct the discretization ensuring the continuity at the nodal level; on the other hand, coarse fineness is observed in the connection areas).

Setting finite element dimensions on contact surfaces

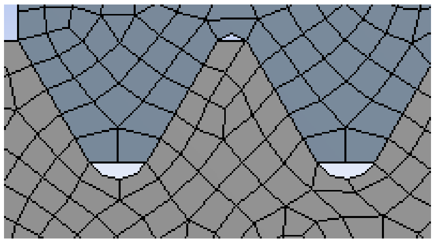
↳ Mesh → ↲Insert → ↲ Sizing → Details of "Sizing" - Sizing, **Scope**: ↲Geometry → ↳ Piulita → ↲ Hide Body → ↲ → [select with Ctrl + ↲ the thread lines of the screw threads] → ↲ Apply; Details of "Edge Sizing" - Sizing: ↲Element Size, [input value, 1,0] → ↳ Piulita → ↲ Show Body → ↳ Surub → ↲ Hide Body → ↲ → [select with Ctrl + ↲ the thread lines of the nutthreads] → Details of "Edge Sizing 2" - Sizing : ↲Element Size, [input, 1,0]; ↳ Surub → ↲ Show Body.

Automatic remeshing

↳ Mesh → ↲ Generate Mesh (fig. c, d).



c.

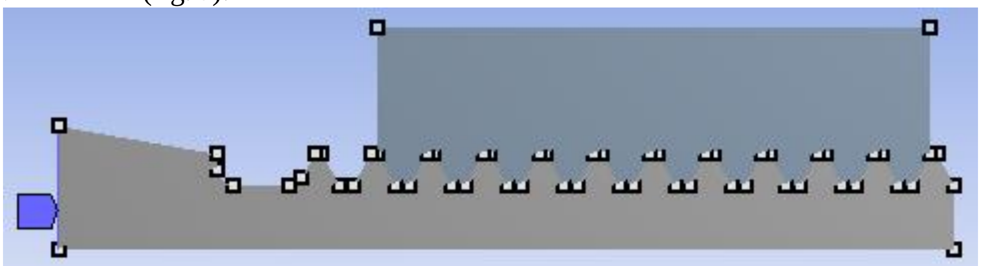


d.

C.5 ***Supports and restraints modelling***

Fixed support constraint (cancels all 6 degrees of mobility)

Outline: ↲ Static Structural (A5) → ↲ Supports → ↲ Fixed Support; ↲ **Model (A4)** → ↲ → [select with ↲ the edge (fig. a)]; ↲ Fixed Support → Details of "Fixed Support", **Scope**: ↲Geometry → ↲ No Selection → ↲ Apply (fig. a).

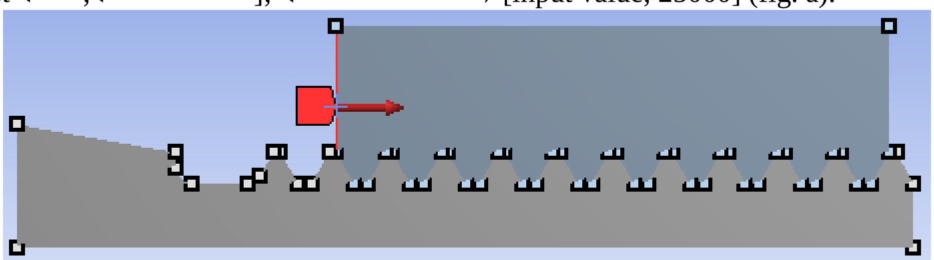


a.

C.6 ***Loads modelling***

Distributed force load on one edge

Outline: ↳ Static Structural (A5) → ↲Insert → ↲ Force → Details of "Force", **Scope**: ↲Geometry → ↲ → [will be selected with ↲ the edge on which the force is applied] → ↲ Apply; **Definition**: ↲Define By → [select from the list ↲, ↲Components]; ↲X Component → [input value, 25000] (fig. a).



a.

D. SOLVING THE FEA MODEL

D.1 Setting the convergence criterion for solving the nonlinear physical model (with friction)

, Outline: → ↵ ⊞ Solution (A6) → ↵ Solution Information, Details of "Solution Information",

↵ ⊞ Solution Information: ↵ Solution Output → [select from list ↵, ↵ Force Convergence] (the force convergence criterion is adopted).

D.2 Setting the results

Setting the total displacement

, Outline: ⊞ Solution (A6) → ↵ Insert → ↵ Deformation → ↵ Total.

Setting the equivalent stress

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Equivalent (von-Mises).

Setting the normal axial stress

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Normal → Details of "Normal Stress", Scope: ↵ Orientation → [select from list ↵, ↵ X Axis];

Setting the normal radial stress

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Normal → Details of "Normal Stress", Scope: ↵ Orientation → [select from list ↵, ↵ Y Axis];

Setting the normal tangential stress

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Normal → Details of "Normal Stress", Scope: ↵ Orientation → [select from list ↵, ↵ Z Axis];

Setting the structural error

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Stress → ↵ Error.

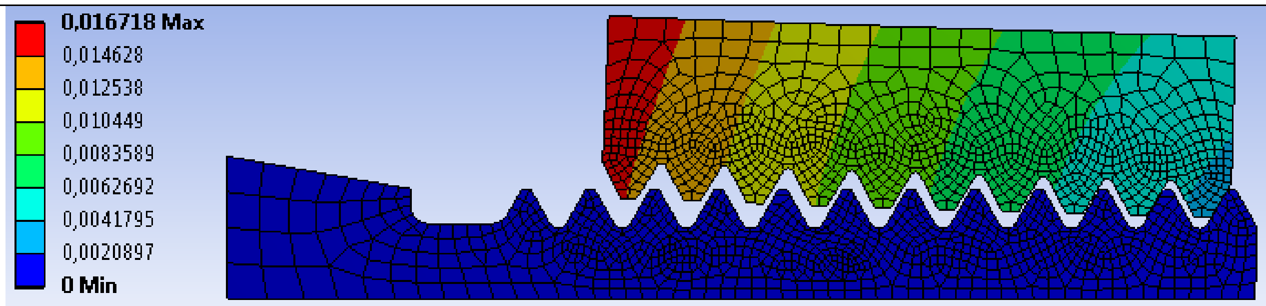
D.3 Launching the solving module

, Outline: ↵ ⊞ Solution (A6) → ↵ Solve.

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

, Outline: ↵ ⊞ Solution (A6) → ↵ Total Deformation (fig. a); ↵ → [select from list with ↵, ↵ Contour Bands] (visualization of contours; ↵ → [select from list with ↵, ↵ Show Elements] (visualization the FE structure); ↵ Result → [select from list with ↵, ↵ 97 (0.5x Auto)] (select the scale factor); Graph → ↵ Animation (view the animation).

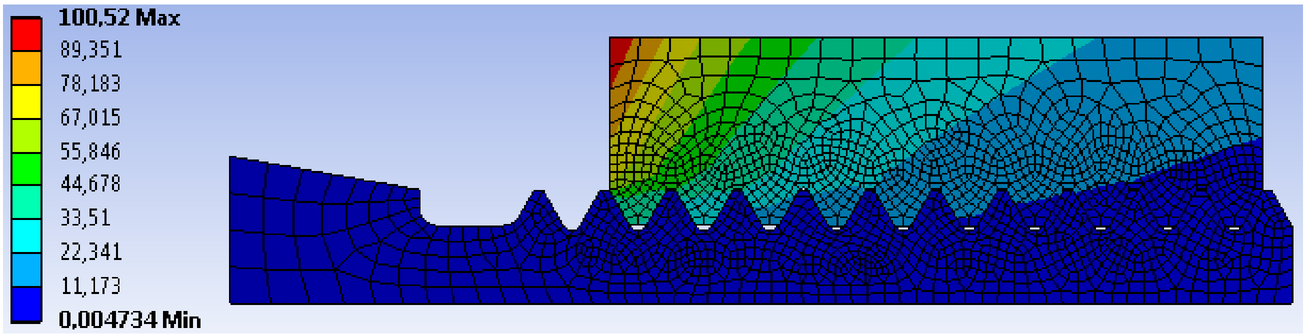


a.

E.2 Viewing the stresses field

<u>*Viewing the equivalent stress field*</u>

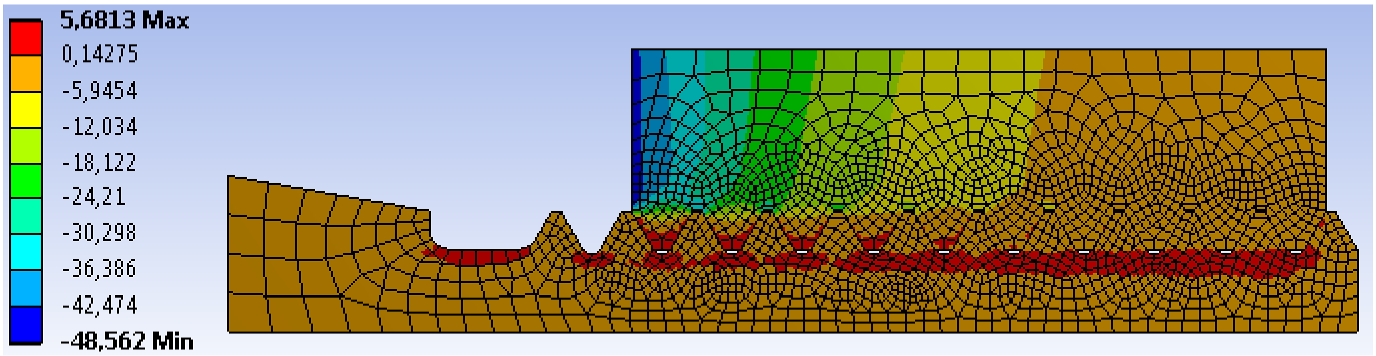
, Outline: ↵⊞···? Solution (A6)→↵···✓ Equivalent Stress (fig. a) ; Graph → ↵ Animation ▶ ■ (view the animation); ↵Result → [select from list with ↵▼, ↵1.0 (True Scale)] (select the scale factor);



a.

<u>*Viewing the axial stress field*</u>

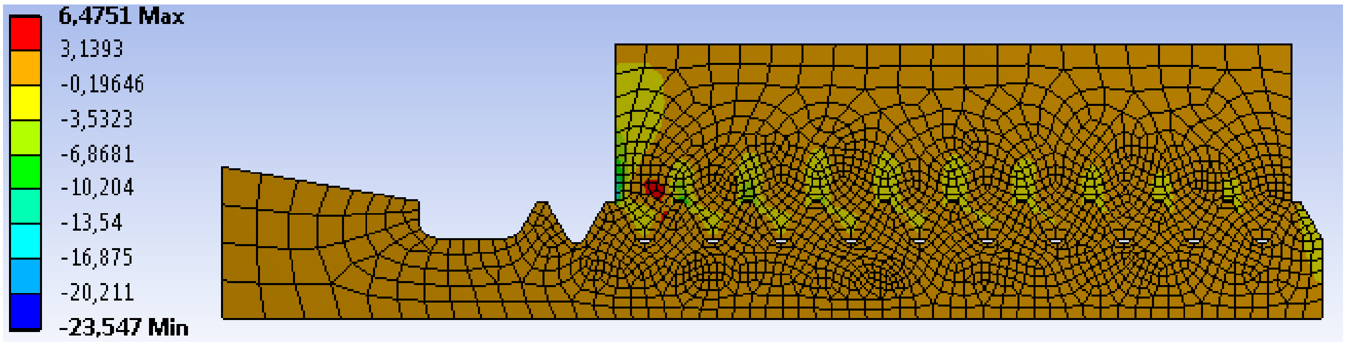
, Outline: ↵⊞···? Solution (A6)→↵···✓ Normal Stress (fig. b).



b.

<u>*Viewing the radial normal stress field*</u>

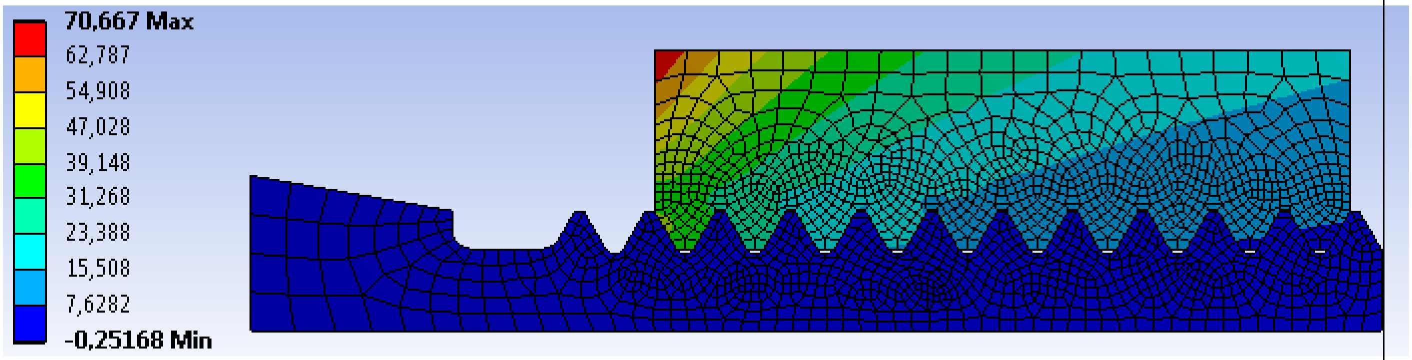
, Outline: ↵⊞···? Solution (A6)→↵···✓ Normal Stress 2 (fig. c).



c.

Viewing the tangential normal stress field

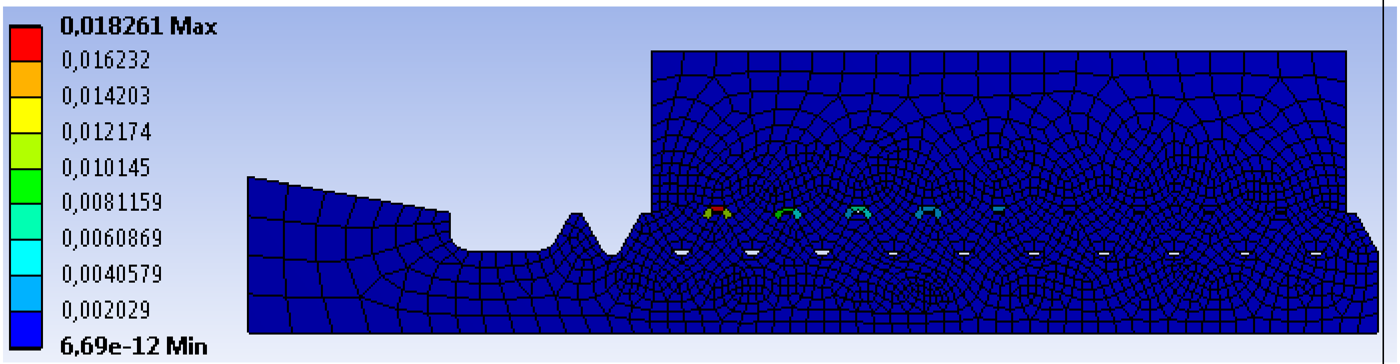
, Outline: Solution (A6) → Normal Stress 3 (fig. d).



d.

E.3 Viewing the structural error

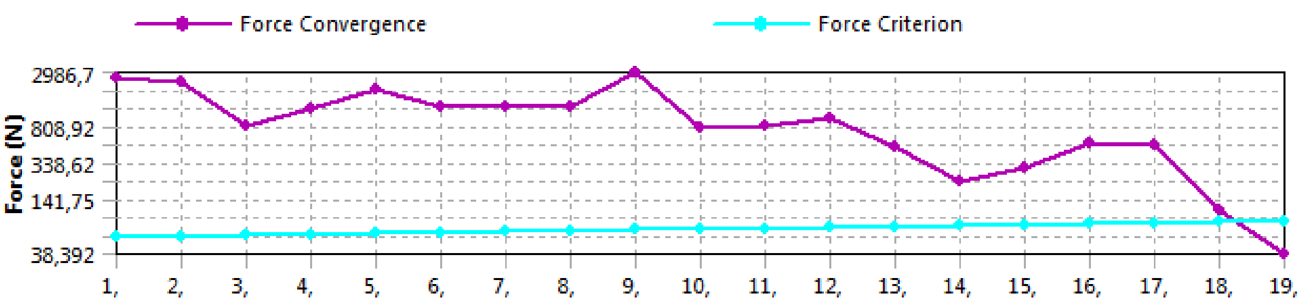
, Outline: Solution (A6) Structural Error (fig. a).



a.

E.4 View the convergence graph of the solution of the nonlinear problem

, Outline: Solution Information (fig. a).



a.

F. ANALYSIS OF RESULTS

F.1 Interpretation of results

Following the analysis of the results obtained, as a result of the modeling and post-processing of the results (subchapter E), the following are highlighted:

- Following the deformation process of the semi-finished product as a result of the action of the force (subchapter B2 fig. a) there are increased displacements (max. 0.016718 mm, subchapter E.1) in the area of action of the load (bearing of the nut).

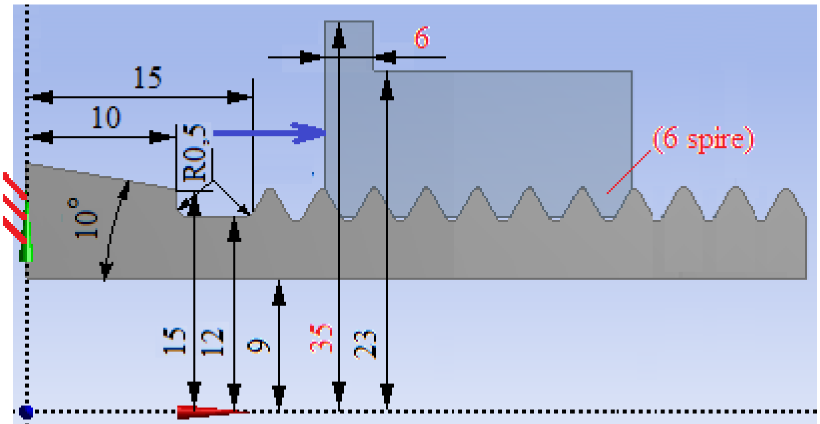
- The equivalent stress has increased values (max. 100.52 MPa; subchapter E.2, fig. a) in the nut body in the bearing area on hub 1 (subchapter A.2, fig. a); following the distribution of the equivalent tension in the threaded areas, the almost same stress of the last 3-4 pairs of turns is observed (subchapter E.1, fig.a), which shows that the force is transmitted, mainly, by these turns (situation verified by experiments).
- From the analysis of the axial tension (subchapter E.2, fig. b) the compression request of the nut body with maximum value (-48.562 MPa) and the tension request with lower values in the screw body are highlighted.
- Normal radial stresses, especially compression, have low values (subchapter E.2, fig. c)
- In the subchapter. E.2, fig. b highlights the tensile stress with increased values (70,667 MPa) of the tangential (circumferential) stresses in the outer area of the nut and the compression stress with much lower values in the screw body.

F.2 Analysis of the precision and convergence of solving the nonlinear model

The much reduced values of the structural error field (max 0.01826 mJ, subchapter E.3) indicate that the stress values are close to the exact ones. In addition, from subchapter. E.4 highlights the fast convergence (19 pitches) of the model solving algorithm and the calculation time is reduced.

F.3 Design studies

From the analysis of the above results, two negative aspects of the screw-nut structure can be synthesized: the uneven distribution of the axial load on the pairs of turns in contact (out of 10 pairs of turns, only 3-4 are active); increased stresses occurring in the nut body, especially in the bearing area on hub 1 (subchapter fig. a). Starting from the fact that the tensions in the thread and the body of the screw have low values (subchapter E.2) in order to diminish the two negative aspects, dimensional and / or nut shape changes are required. Thus, two options are proposed for optimizing the shape of the nut. The first involves increasing the outer diameter of the nut from 23 mm to 30 mm and reducing its length to 6 turns (subchapter A.2, fig. a). The second variant proposes stiffening the nut in the bearing area by inserting a collar in the bearing bearing area and reducing the length of the nut to 6 turns.



In the case of the proposed variants, it is necessary to modify the analysis model and solve it by going through the successions: DM, Tree Outline: …… change size value ….. ↵ Generate; M, Outline: ↳ ⊞ Geometry → ↵ Refresh Geometry; ↵ Solve. After solving the model, the results are reanalyzed and reinterpreted.

G. CONCLUSIONS

In this paper, the modeling and analysis with finite elements were also made for teaching purposes following the user's initiation with the main stages of developing an FEA application in ANSYS Workbench, which emphasizes, in particular, the modeling and analysis of a deformable element of its contacts with another adjacent element.

The adopted FEA model involves considering the multiple friction contacts of a screw-nut threaded assembly of linearly behaved materials. For the analysis, a symmetrical axial plane geometric model (2D) with line-to-line contact connections was developed. External loading was performed by means of a force distributed on a line. As a result of solving the model with nonlinear finite elements adopting the method of force convergence, results were obtained with increased precision, the values of the obtained parameters (displacements, stresses, structural error) being useful for optimizing the shape and dimension of the nut element.

Application: FEA-A.9
Tight assembly on the cone

KEY WORDS
Nonlinear static analysis, Axial-symmetric stress state, Linear material, 2D geometric model, 2D finite element, Linear finite element, Mechanical friction contact, Structural error, Tight assembly on the cone, Mechanical subassembly

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

FEA, as a general method of studying the physical phenomena and processes in mechanical structures, also allows the analysis of mechanical fields that occur in the case of *mechanical assembly contacts* that involve consideration of elastically deformable surfaces in direct contact and sliding friction between them.

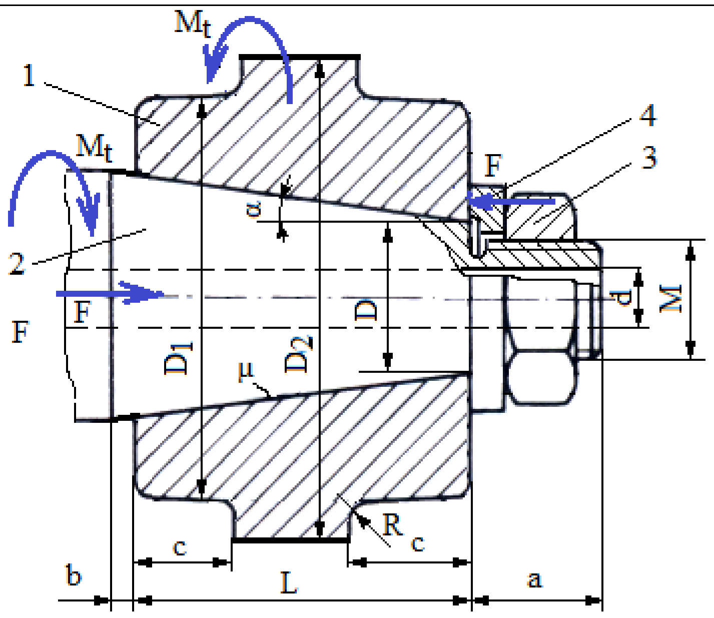
Tight assembly on the cone frequently used in the construction of mechanical systems form complex spatial structures involving mechanical frictional contacts that participate in load transmission. Starting from the fact that these structures cannot be accurately analyzed with classical theoretical and / or experimental methods, this problem is further treated by modeling and FEA.

A.2 Application description

The tight assembly on the cone transmits the frictional torque M_t from the hub 1 to the shaft 2 (fig. a). For this, it is necessary to carry out the axial tightening by developing a pressing force F by tightening the nut 3 in relation to the external thread applied on the shaft 2. The tight assembly on the cone is described by the minimum inner diameter of the bore D, the angle of inclination of the generator, α, and the length of the hub, L. The shaft with conical surface on the outside has an axial hole with diameter d. For FEA, the assembly is considered the hub with a conical bore on the inside as the inside of a wheel (toothed, belt, chain), which has on the outside two lateral cylindrical sections identical in diameters and lengths, D1 and, respectively, c and a central section with a diameter a portion of a wheel disc.

A.3 The application goal

For the analysis of the displacement and tension fields in the assembly area taking into account the friction between the shaft and the hub ($\mu = 0.2$) it is considered (subchapter A.2. Fig. A): D = 30 mm, $\alpha = 10^o$, d = 18 mm, D1 = 50 mm, D_2 = 80 mm, a = 30 mm, b = 10 mm, c = 12 mm, M = 30 mm, L = 35 mm. In order to transmit the torque M_t, the axial force load, F = 45000 N, is required by means of the screw-nut threaded assembly. The shaft and hub are made of construction steel without heat treatment (eg E235).



a.

B. THE FEA MODEL

B.1 The model definition

In order to design the FEA model of the nut / screw in interaction, it is necessary to consider two adjacent areas of the two elements adopting the following simplifying hypotheses:

- considering that there are no significant variations on the circumference of the physical parameters (displacements and stresses), a planar model can be adopted that can be framed in the axial-symmetrical state of stresses.
- existing friction in mechanical contacts,
- adoption of material strength constraints (embedding, action of force distributed on the surface),
- the material has an elastic linear behavior,

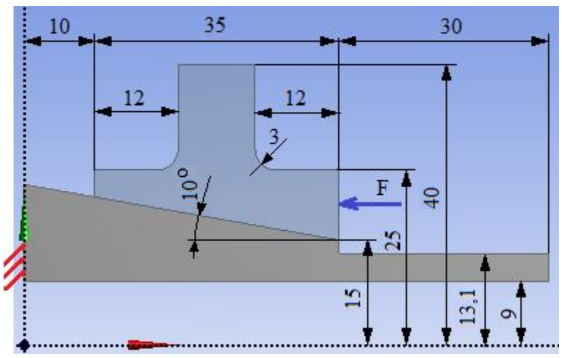
the deformation takes place statically (the variation of the deformation force over time is not taken into account).

B.2 The analysis model description

To simulate the behavior of the tight assembly on the cone, the axial section with the dimensions of fig. a. The threaded and connecting area of the shaft head portion is neglected and is considered to be cylindrical with a diameter of 26.2 mm.

For analysis, the structure is considered axial-symmetrical and is modeled with 2D finite elements. In order to simulate the behavior of the assembly as close as possible to reality, the friction between the assembled elements will be taken into account, the coefficient of friction μ = 0.2.

The loading will be done on the front surface of the nut with F = 15000 N.



a.

B.3. Characteristics of the material and the environment

For linear static analysis the following resistance characteristics of E335 material are considered:

- longitudinal modulus of elasticity, E = 206000 N / mm^2;
- Poisson's ratio, ν = 0,3.

Average working temperature of the subassembly, T_0 = 20 ° C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project

WB, Toolbox: ↲⊟ Analysis Systems → ↲↲ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural în *FEA-A.9*].

Problem type setting (2D)

A : ↳ Geometry → ↲Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↲Analysis Type, [select from the list ↲▼, ↲ 2D] → [close the window, ↲X].

Saving of the project

↲ Save As... → WB Save As, File name: [input name, *FEA-A.9*] → ↲ Save.

C.2 Modelling of material and environment characteristics

WB, Project Schematic: ↳ Engineering Data ✓ → ↲ Edit... → Outline of Schematic A2: Engineering Data: ↲ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, Young's Modulus, [select from column C (Unit) cu / with ↲▼, ↲MPa], [input in column B (Unit) valoarea / value, 206000] → ↲ Update Project → ↲ Return to Project (the other parameters remain the default).

C.3. Geometric modeling

C.3.1 Model loading, DesignModeler (DM)

WB, Project Schematic: ↳ Geometry → ↲DM New Geometry... → ANSYS Workbench: ↲ Millimeter, ↲ OK.

C3.2 Sketch generation 1 (shaft)

Viewing default plane (XY)

DM, Tree Outline: ↲Sketching → (Look at face/Plane/Schetch), [automatically view of default plane XY Plane];

Generating of sketch 1

Polyline generation

↲Draw → ↲ Polyline → [the polyline will be drawn by marking with ↲ the points respecting the restrictions of coincidence C, of horizontality H and verticality V (the last point overlaps over the first, coincidence restriction P]→

→ [will be selected with a point in the graphics area] (context menu appears) → ↲Closed End (fig. a).

Inclined line split

↲Modify → ↲ Split → [will be marked with ↲ the point on the inclined line] (fig. b).

Sketch dimensioning

Dimensioning in the horizontal direction

Sketching Toolboxes: ↲Dimensions → ↲ Horizontal → [select with ↲ the lines parallel to the Y axis] (the dimension is automatically displayed) → Details View, Dimensions: , ☐ H → [input value, 10/30/75] (fig. b).

Dimensioning in the vertical direction

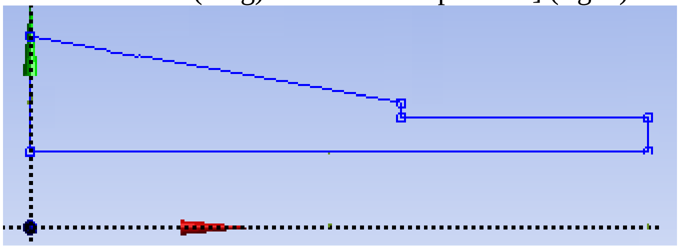
↲ Vertical → [select with ↲ the lines parallel to the X axis] (the dimension is automatically displayed) → Details View, Dimensions: : ↲☐ V → [input value, 15/13,1/9 fig. b).

Dimensioning the angles

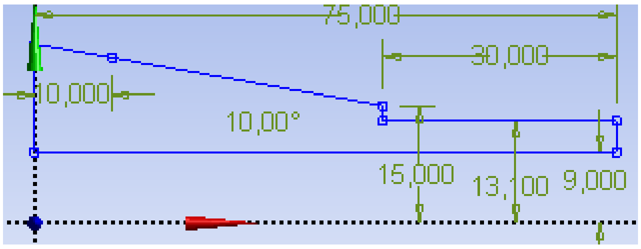
↲ Angle → [select with ↲ angle lines] (automatically view dimension) → Details View, Dimensions: : ↲☐ A → [input value, 10 fig. b).

Edit dimensions

↵ Display → ↵ Name: ☑ (dezactivate) → ↵ Name: ☐ Value: ☑ (activate); ↵ Move → [select the dimension with ↵ and move (drag) to the desired position] (fig. a).



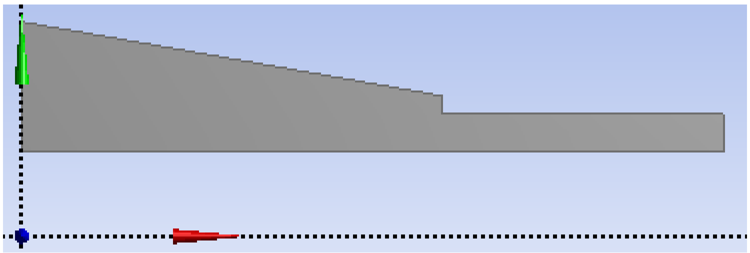
a.



b.

C.3.3 Generation of the shaft surface

DM: ↵ Concept → ↵ Surfaces From Sketches → ↵ Sketch1 → Details View, Details of Surface: ↵ Base Objects → ↵ Apply; ↵ Generate (generate surface, fig. a); ↵ Sketch1 → ↵ Hide Sketch; ↵ Surface Body → Details View, Details of Surface Body: ↵ Body, [input name, Arbore / Shaft].



a.

C.3.4 Sketch generation 2 (hub)

2nd sketch initialization

DM: ↵ (New Sketch) → (the object is automatically indexed in the specification tree Sketch2).

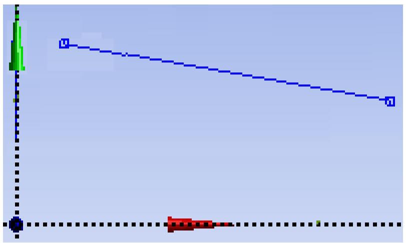
Contour generation

Activare schiţă 1 / 1st sketch activation

Tree Outline: ↵ Sketch1 → ↵ → ↵ (Display Model).

Common line generation

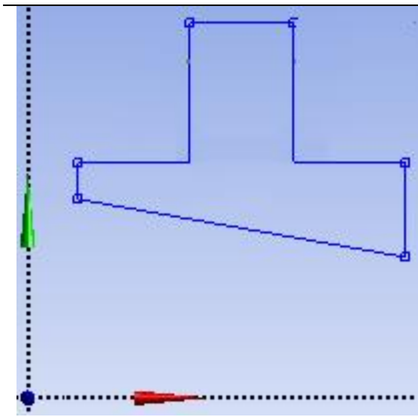
↵ Sketching → Sketching Toolboxes: ↵ Draw → ↵ Line → [select with ↵ the end points of the common line respecting the coincidence conditions P] (fig. a).



a.

Hub contour generation

↵ Polyline → [draw the polyline by ↵ marking the points associated with the hub body respecting the restrictions of coincidence with a point (P), verticality (V) and horizontality (H)] (fig. b).



b.

1st sketch masking

↵ Modeling → Tree Outline: ↵ Sketch1 → ↵ Hide Sketch → ↵ (Display Model).

Fillet generation

↵ Sketching → Sketching Toolboxes: ↵ Modify → ↵ Fillet → Radius:, input value of radius, 3] → [is marked with ↵ the pairs of lines to be connected] (fig. c).

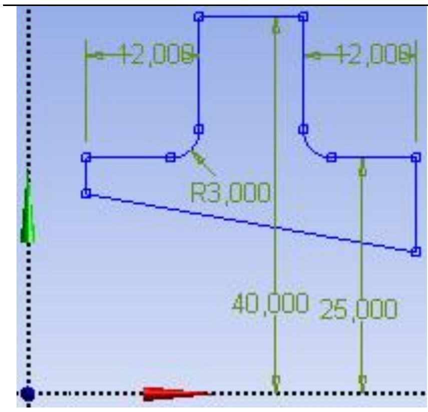
Contour dimensioning

Dimensioning in the horizontal direction

Sketching Toolboxes: ↵ Dimensions → ↵ Horizontal → [select with ↵ the lines parallel to the Y axis] (the dimension is automatically displayed) → Details View, Dimensions: , ☐ H → [input value, 12/12] (fig. c).

Dimensioning in the vertical direction

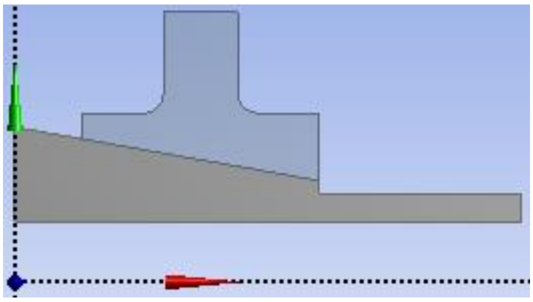
↵ Vertical → [select with ↵ the lines parallel to the X axis] (the dimension is automatically displayed) → Details View, Dimensions: ↵ V → [input value, 40/25] (fig. c).



c.

C.3.5 Hub surface generation

DM: ↵ Concept →
↵ Surfaces From Sketches → ↵ Sketch2 → Details View, Details of Surface: ↵ Base Objects → ↵ Apply; ↵ Generate (generate surface, fig. a); ↳ Sketch1 → ↵ Hide Sketch. ↵ Surface Body → Details View, Details of Surface Body: ↵ Body, [input name, Butuc / Hub].



a.

C.3.6 Saving of geometric model

DM: ↵ (Save Project) → ↵ (Close).

C.4. Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

Launching of the finite element modeling module

WB, Project Schematic: ↳ Model → ↵ M Edit... → [launch module *Mechanical [ANSYS Multiphysics]*.

Setting the unit of measure system

M: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

Setting the material characteristics

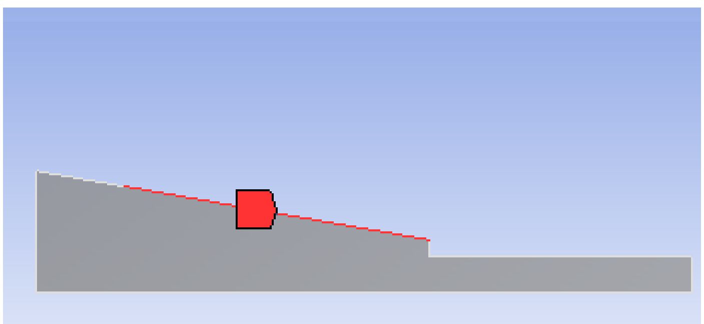
Outline: ↵ ⊞ Geometry → ↵ Surub → Details of "Surub", Material: ↵ Assignment → [select from the list ↵, ↵ Structural Steel (default)]; ↵ Piulita → Details of "Piulita", Material: ↵ Assignment → [select from the list ↵, ↵ Structural Steel (default)];

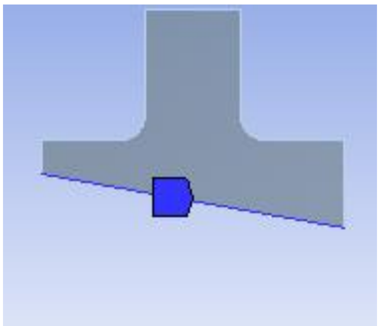
Setting the model type (axial asymmetric)

Outline: ↵ ⊞ Geometry → Details of "Geometry", Definition: ↵ 2D Behavior, [select from the list ↵, ↵ Axisymmetric].

C.4.2 Modelling the friction connections

M, Outline: ↵ Connections → ↵ Insert → ↵ Manual Contact Region → Details of "Bonded - No Selection To No Selection", Definition: ↵ Type → [select from the list ↵, ↵ Frictional]; ↳ Butuc → ↵ Hide Body → ↵ →[select with ↵ the inclined line of the shaft, fig. a] → Details of "Frictional - Arbore To Butuc", Scope: ↵ Contact → ↵ Apply (option Contact Bodies will index automatically, Arbore); ↳ Butuc → ↵ Show Body → ↳ Arbore → ↵ Hide Body → ↵ → [the inclined line of the hub, fig. b] → Details of "Frictional - Arbore To Butuc", Scope: ↵ Target → ↵ Apply (option Target Bodies will index automatically, Butuc); ↵ Definition: ↵ Behavior → [select with ↵, ↵ Symmetric]; Friction Coefficient → [input value, 0,2]; ↵ **Advanced** → ↵ Formulation → [select with ↵ ▸, ↵ Augmented Lagrange] (the method of solving the nonlinear model); ↳ Arbore → ↵ Show Body .



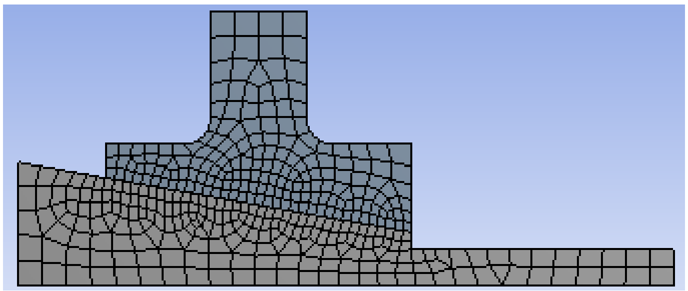


a. ***b.***

C.4.3 Model discretization and analysis type setting

Automatic meshing

, Outline: ↳ Mesh → ↵ Generate Mesh (fig. a, b).



a.

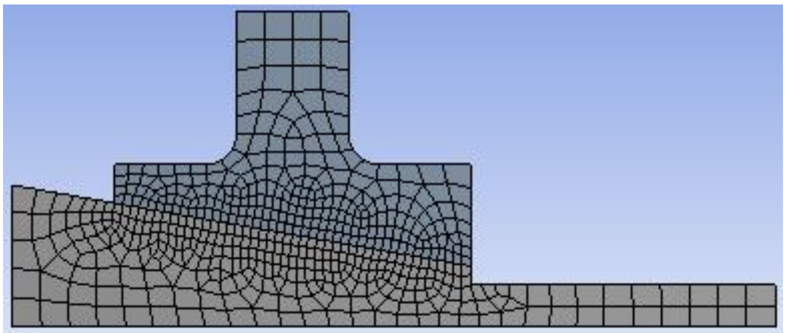
Obs. In fig. a, there are discontinuities of the finite element structure at the level of the contact surface, which leads to an inadequate modeling of the contact phenomena and it is necessary to correct the discretization ensuring the continuity at the nodal level.

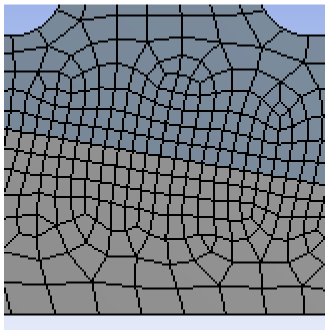
Setting finite element dimensions on contact surfaces

↳ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, **Scope**: ↵ Geometry → ↳ Butuc → ↵ Hide Body → ↵ → [select with Ctrl + ↵ the contact line] → ↵ Apply; Details of "Edge Sizing" - Sizing: ↵ Element Size, [input value, 0,8] → ↳ Butuc → ↵ Show Body → ↳ Arbore → ↵ Hide Body → ↵ → [select with Ctrl + ↵ the contact line] → Details of "Edge Sizing 2" - Sizing: ↵ Element Size, [input value, 0,8]; ↳ Arbore → ↵ Show Body .

Automatic remeshing

↳ Mesh → ↵ Generate Mesh (fig. b, c).



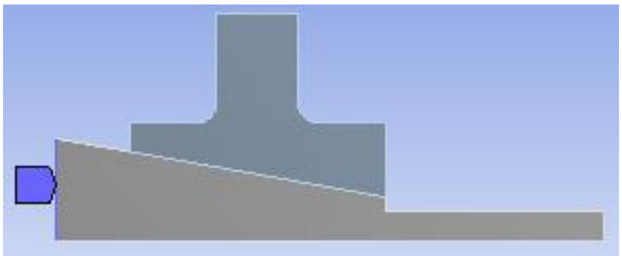


b. ***c.***

C.5. *Supports and restraints modelling*

Fixed support constraint (cancels all 6 degrees of mobility)

Outline: ↲ Static Structural (A5) → ↲ Supports → ↲ Fixed Support; ↲ Model (A4) → ↲ → [select with ↲ the edge (fig. a)]; ↲ Fixed Support → Details of "Fixed Support", Scope: ↲ Geometry → ↲ No Selection → ↲ Apply (fig. a).

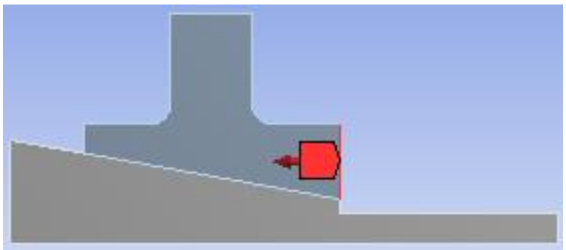


a.

C.6 Load modeling

<u>*Distributed force load on one edge*</u>

Outline: ↳ Static Structural (A5) → ↲ Insert → ↲ Force → Details of "Force", Scope: ↲ Geometry → ↲ → [will be selected with ↲ the edge on which the force is applied] → ↲ Apply; Definition: ↲ Define By → [select with ↲, ↲ Components]; ↲ X Component → [input value, -45000] (fig. a).



a.

D. SOLVING THE FEA MODEL

D.1 Setting the convergence criterion for solving the nonlinear physical model (with friction)

Outline: → ↲ Solution (A6) → ↲ Solution Information, Details of "Solution Information", ↲ Solution Information: ↲ Solution Output → [select from list ↲, ↲ Force Convergence] (the force convergence criterion is adopted).

D.2 Setting results

<u>*Setting the total displacement*</u>

Outline: ↳ Solution (A6) → ↲ Insert → ↲ Deformation → ↲ Total.

<u>*Setting the equivalent stress*</u>

↳ Solution (A6) → ↲ Insert → ↲ Stress → ↲ Equivalent (von-Mises).

<u>*Setting the normal axial stress*</u>

↳ Solution (A6) → ↲ Insert → ↲ Stress → ↲ Normal → Details of "Normal Stress", Scope: ↲ Orientation → [select from list ↲, ↲ X Axis];

<u>*Setting the normal radial stress*</u>

↳ Solution (A6) → ↲ Insert → ↲ Stress → ↲ Normal → Details of "Normal Stress", Scope: ↲ Orientation → [select from list ↲, ↲ Y Axis];

<u>*Setting the normal tangential stress*</u>

↳ Solution (A6) → ↲ Insert → ↲ Stress → ↲ Normal → Details of "Normal Stress", Scope: ↲ Orientation → [select from list ↲, ↲ Z Axis];

<u>*Setting the structural error*</u>

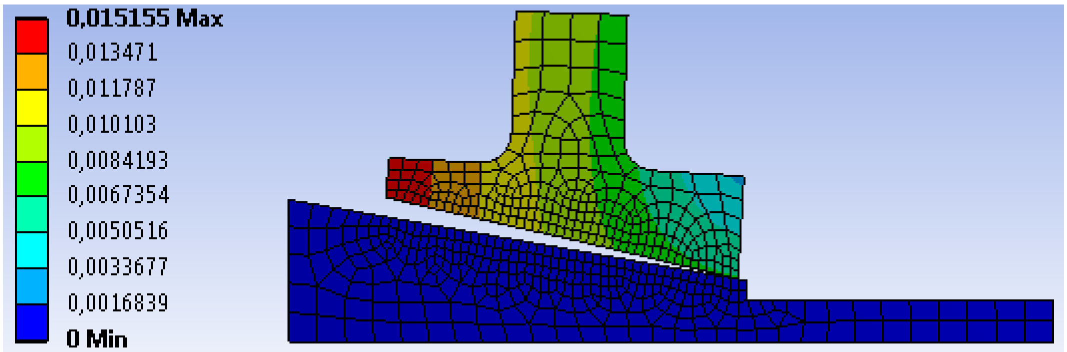
↳ Solution (A6) → ↲ Insert → ↲ Stress → ↲ Stress → ↲ Error.

D.3 Launching the solving module
, Outline: ↲⊞┈? Solution (A6) → ↲ Solve .

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

, Outline: ↲⊞┈? Solution (A6) → ↲┈ Total Deformation (fig. a); ↲ → [select from list ↲▼, ↲ Contour Bands] (vizualization contours in bands); ↲ → [select from list with ↲▼, ↲ Show Elements] (visualization the FE structure); ↲Result → [select with ↲▼, ↲1,3e+002 (0.5x Auto)] (select the scale factor); Graph → ↲ Animation ▶ ■ (view the animation).

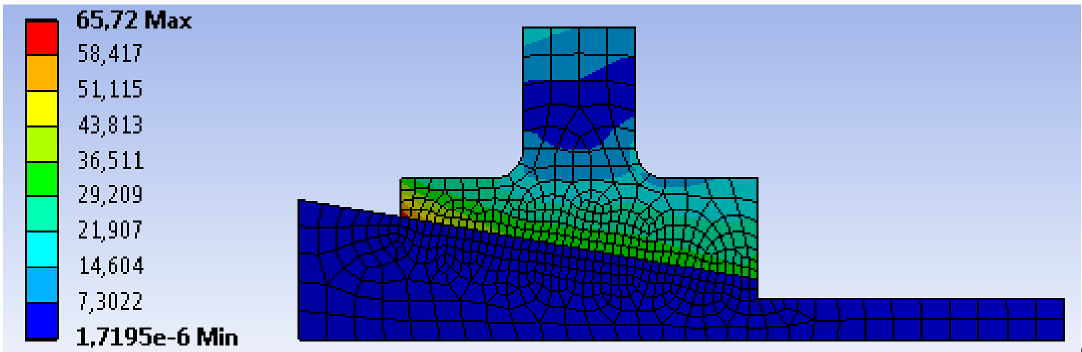


a.

E.2 Viewing the stresses field

Viewing the equivalent stress field

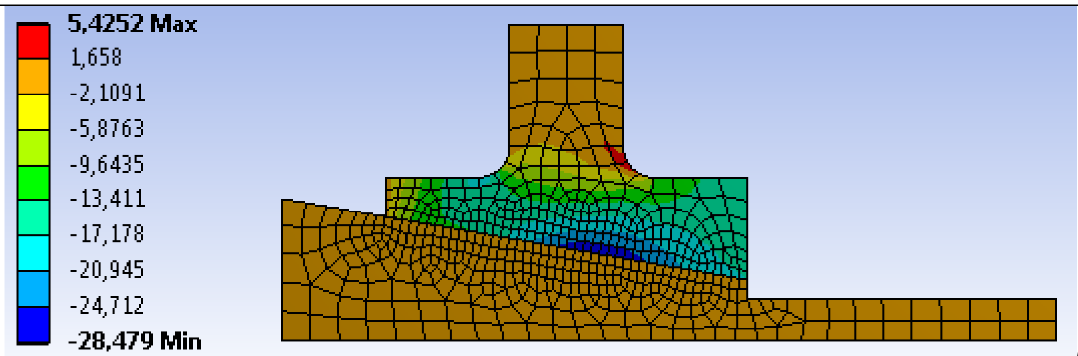
, Outline: ↲⊞┈? Solution (A6) → ↲┈ Equivalent Stress (fig. a) ; Graph → ↲ Animation ▶ ■ (view the animation); ↲Result → [select from list with ↲▼, ↲1.0 (True Scale)] (select the scale factor);



a.

Viewing the axial stress field

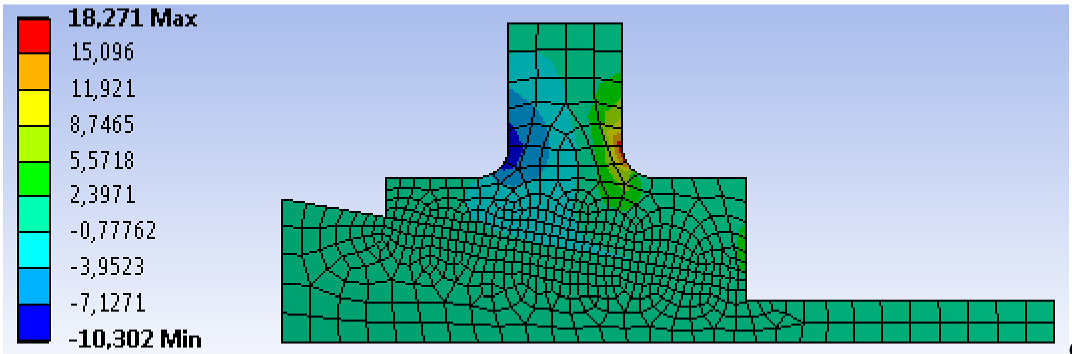
, Outline: ↲⊞┈? Solution (A6) → ↲ ┈ Normal Stress (fig. b).



b.

Viewing the radial normal stress field

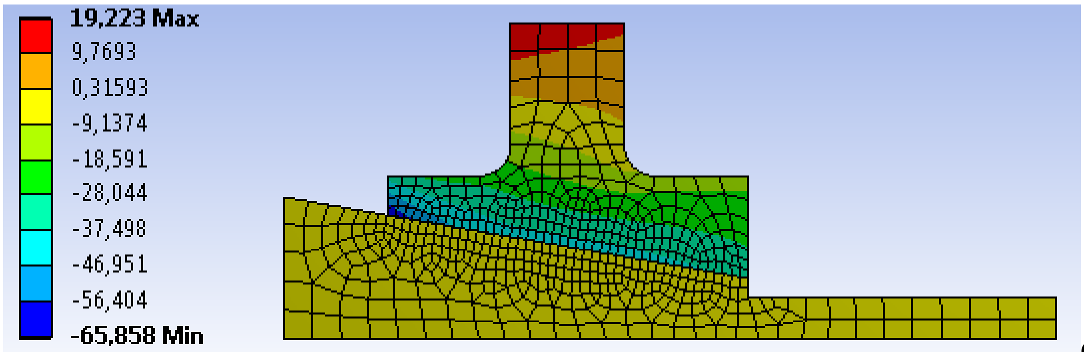
, Outline: ↵⊞ ? Solution (A6) → ↵ Normal Stress 2 (fig. c).



c.

Viewing the tangential normal stress field

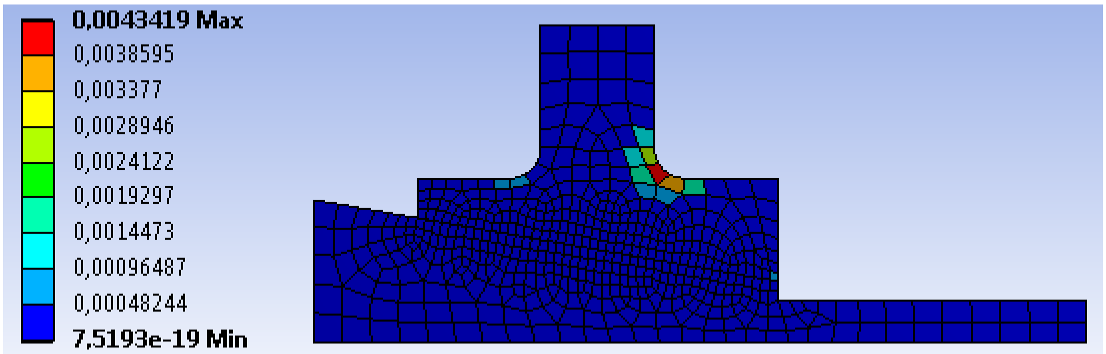
, Outline: ↵⊞ ? Solution (A6) → ↵ Normal Stress 3 (fig. d).



d.

E.3 Viewing the structural error

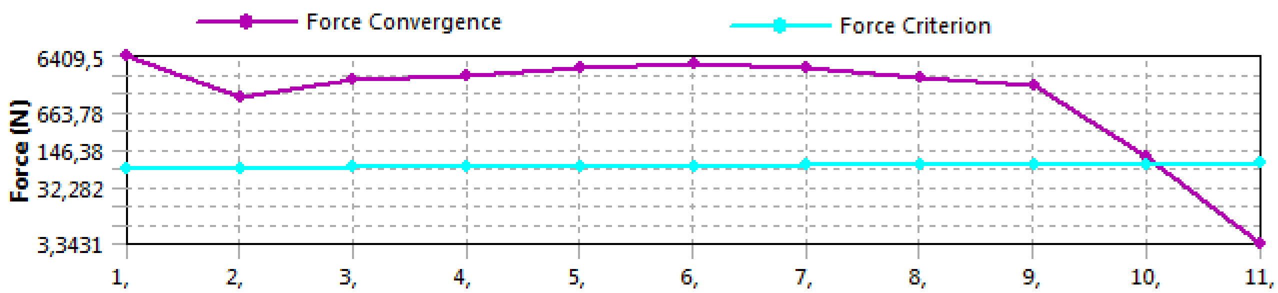
, Outline: ↵⊞ ? Solution (A6) ↵ Structural Error (fig. a).



a.

E.4 Viewing the convergence graph of the solution of the nonlinear problem

, Outline: ↵ Solution Information (fig. a).



a.

F. ANALYSIS OF RESULTS

F.1 Interpretation of results

Following the analysis of the results obtained, as a result of the modeling and post-processing of the results (subchapter E), the following are highlighted:

- Following the process of deformation of the elements of the subassembly as a result of the action of the nut (subchapter A.2, fig. A) there are increased displacements (max. 0.015155 mm, subchapter E.1) in the area of the hub with the maximum diameter of bore.
- The equivalent stress has increased values (max. 65.72 MPa; subchapter E.2, fig. A) in the body of the hub in the area with the maximum diameter of the bore (subchapter A.2, fig. A).
- From the analysis of the axial tension (subchapter E.2, fig. B) the compression request of the hub body with maximum value, -28,479 MPa, and the tension request with low values in the hub in the connection area from the outside are highlighted.
- Normal radial stresses, especially compression, have low values (subchapter E.2, fig. C)
- In subchapter. E.2, fig. d highlights the compression request with increased values (65,858 MPa) of the tangential (circumferential) stresses in the hub in the area with the maximum diameter of the bore and the tension request with much lower values in the hub body.

F.2 Analysis of the precision and convergence of solving the nonlinear model

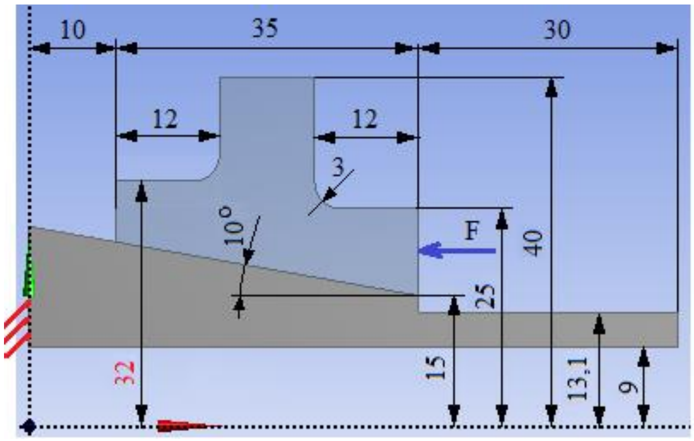
The much reduced values of the structural error field (max 0.0436 mJ, subchapter E.3) indicate that the stress values are close to the exact ones. In addition, from subchapter. E.4 highlights the fast convergence (19 pitches) of the model solving algorithm and the calculation time is reduced.

F.3 Studies for design

From the analysis of the above results, the non-uniformity of the tightening along the conical bore is highlighted and correlated with this increased tensions in the shoulder area of the hub on the left side. In order to reduce these disadvantages, it is proposed to increase the shoulder of the hub on the left side (fig. A). Thus, it is necessary to modify the analysis model and solve it by going through the successions:

DM, Tree Outline: …… modify the value of dimension ….. ↵ Generate; , Outline: ⮡ ⊞ Geometry → ↵ Refresh Geometry; ↵ Solve.

After solving the model, the results are reanalyzed and reinterpreted.



a.

G. CONCLUSIONS

Modeling and analysis with finite elements in this paper were also made for teaching purposes following the user's initiation with the main stages of developing an FEA application in ANSYS Workbench, which emphasizes, in particular, the modeling and analysis of a deformable element and of its contacts with another adjacent element.

The adopted FEA model involves considering the frictional contact of a cone-tightening assembly. For analysis, a symmetrical axial plane geometric model (2D) with line-to-line contact type was developed. External loading was performed by means of a force distributed on a line.

As a result of solving the model with nonlinear finite elements adopting the method of force convergence, results were obtained with increased precision, the values of the obtained parameters (displacements, stresses, structural error) being useful for optimizing the shape and dimensions of the Hub element.

Application: FEA-A.10
Optimizing the solutions

KEY WORDS
Linear Static Analysis, Optimization, Linear Material, 2D Geometric Model, 2D Finite Element, Linear Finite Element, Element, Design Parameters, Status Parameters, Objective Function

CONTENT

A. DESCRIEREA PROBLEMEI / PROBLEM DESCRIPTION

A.1 Introduction

In general, the FEA determines values of the output parameters (deformations, displacements, stresses), depending on the preliminary predefined model parameters. Some FEA have distinct optimization modules that for a preliminarily analyzed structure allow the determination of independent parameters, consequence of solving an optimization model that involves minimizing / maximizing some purpose functions while imposing restrictions of other dependent parameters (see Chapter F)

A.2 Application description

For the elaboration of the model of constructive optimization of the fixed bar and loaded with the force F from fig.a consider:

- predefined parameters: L, a, G;
- design parameters (input): D, H;
- state parameter (output): σ_{ech} (equivalent von Mises stress).

Optimization model (see Chapter F)

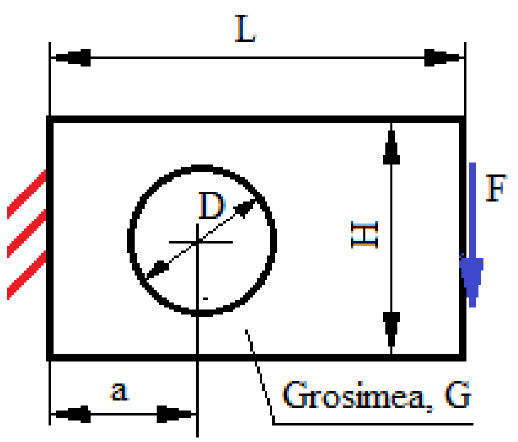
Restrictions:

$D_{min} < D < D_{min}$;

$H_{min} < H < H_{min}$;

$\sigma_{ech} < \sigma_a$ (the permissible stress imposed).

Objective function: Mass → min.



A.3 The application aim

This application presents, using finite element analysis, the algorithm for solving the problem of dimensional constructive optimization of the structure in the figure above. For preliminary FEA we consider: L = 50 mm, H = 40 mm, G = 10 mm, a = 20 mm. The values of the optimization model parameters are: D_{min} = 14 mm, D_{max} = 18 mm, H_{min} = 35 mm, H_{min} = 44 mm, σ_a = 140 Mpa.

B. THE FEA MODEL

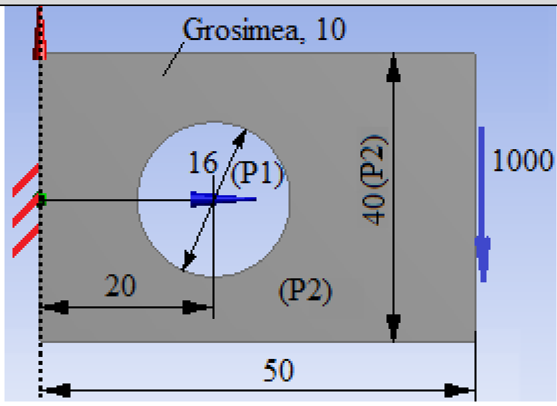
B.1 The model definition

For the analysis and optimization with FE, the following simplifying hypotheses are adopted:

- linear behavior of the material,
- adopting constraints associated with symmetry properties,
- external load by force distributed on the surface,
- the proposed problem is solved in two stages: structural analysis and optimization.

B.2 The analysis model description

Figure a shows the FEA and optimization model associated with the geometric plane model considered in the XY plane. The X-axis is the axis of symmetry of this model. In addition, the design parameters: hole diameter (P1) and width (P2) are also highlighted for optimization.



B.3 Characteristics of the material and the environment

The strength characteristics of E335 material for finite element analysis are:

- the modulus of longitudinal elasticity, E = 210000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.3.
- Average working temperature of the subassembly, T_0 = 20^0 C.

C. PREPROCESSING OF FEA MODEL

C.1 Activarea şi salvarea proiectului / Creating, setting and saving the project

Creating of the project

Toolbox: ↲⊟ Analysis Systems: ↲↲ Static Structural (the subproject window appears automatically - the name can be changed to *Optimization*).

Setting the problem type (3D)

A: ↳ Geometry → ↲Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↲Analysis Type, [select from list ↲▼, ↲3D] → [close the window, ↲✕].

Saving of the project

↲ Save As... → Save As, File name: [input name, FEA-A.10] → ↲ Save.

C.2 Modelling of material and environment characteristics

, Project Schematic: ↳ Engineering Data ✓ → ↲ Edit... → Outline of Schematic A2: Engineering Data:

↵ Structural Steel Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [select from list, C column(Unit) cu ↵▼, ↵MPa], [input in the box in column B (Unit) value, 210000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

C.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

↵↵ Geometry → ↵ New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵ OK.

C.3.2 Sketch generation

Viewing default plane (XY)

DM, Tree Outline: ↵Sketching → (Look at face/Plane/Schetch), [automatically view of default plane XY Plane]

Generating of rectangle

↵ Draw → ↵ Rectangle → [the rectangular line is generated in quadrants I and II by marking ↵ the corner on the Y axis (the coincidence symbol C appears) and the release ↵ in the opposite corner in the other quadrant, fig. a].

Generating of circle

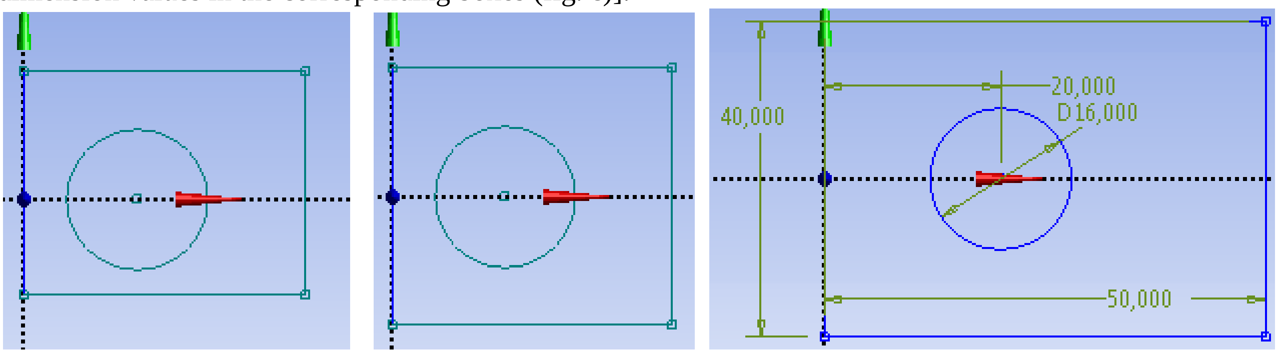
↵ Circle → [a circular line is generated by marking ↵ the center on the X axis (the coincidence symbol C appears) and the release ↵ on the contour, fig. a].

Symmetry constraint

↵Modify → ↵ Constraints → Symmetry → [is activated / deactivated with ↵ option Ignore Axis (☑/☐]) → [is marked with ↵ the X axis of symmetry and with Ctrl + ↵ the horizontal lines] (the symmetrical model with respect to the X axis appears automatically, fig. b).

Dimensioning the sketch

↵ Dimensions → ↵ Semi-Automatic → [is activated with ↵ dimensions] → Details View, ⊟ Dimensions: 4 : [enter the dimension values in the corresponding boxes (fig. c)].



a. **b.** **c.**

C.3.3 Generating the solids object

DM: ↵ Extrude → Details View, ⊟ Details of Extrude1: Geometry, → ↵ Sketch1 → ↵ Apply; ☐ FD1, Depth (>0), [input the depth, 10)]. ↵ Generate

C.3.4 Saving of geometric model

DM: ↵ (Save Project) → ↵ (Close).

C.4. Finite element modelling

C.4.1 Launching of the finite element modeling module

Launching of the finite element modeling module

WB, Project Schematic: ↳ Model → ↵ Edit... → [launching *Mechanical [ANSYS Multiphysics*].

Setting the unit of measure system

M: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (setting the UM system).

Setting the material characteristics

Outline: ↵⊞ Geometry → ↵ Solid → Details of "Solid", Material : ↵Assignment → [select from list ↵ ▸, ↵ Structural Steel (default)];

C4.2 Meshing the model

Mesh → Generate Mesh (meshing with default parameters, fig. C.6,a).

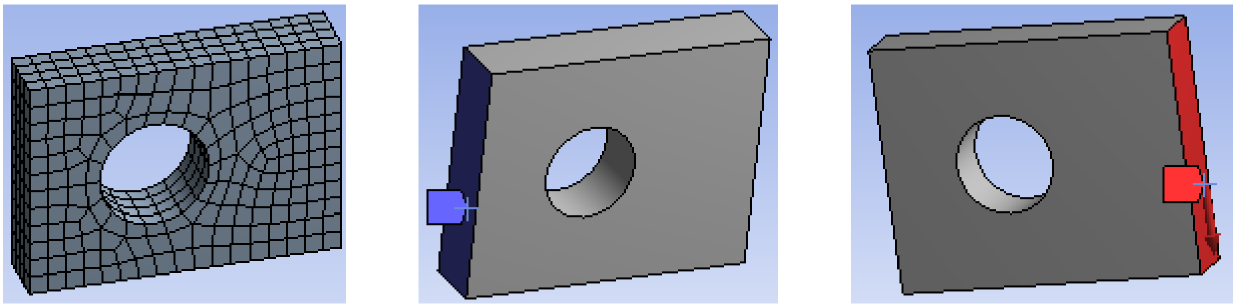
C.5 *Supports and restraints modelling*

Generation of embedding constraint

↵ Static Structural (B5) → ↵ Supports → ↵ Fixed Support; → ↵ (the line selection filter is activated) → [select left face (fig. C.6,b)];

C.6 Modelarea încărcării / *Loads modelling*

↵ Static Structural (B5) → ↵ Loads → ↵ Force ; → ↵ (the line selection filter is activated) → [select right face (fig. c)]; , Details of "Force", Scope: Geometry → [select right face (fig. b)] → ↵ Apply; ↵ Define By, [select from list ↵, ↵Components ; Y Component, [input Y value, -1000];



a. *b.* *c.*

D. SOLVING THE FEA MODEL

D.1 Setting the results

Selecting the total displacement

, Outline; Solution (A6) → ↵ Insert → ↵Deformation → ↵ Total.

Setting the equivalent stress

Solution (A6) → ↵ Insert → ↵ Stress → ↵ Equivalent (von-Mises).

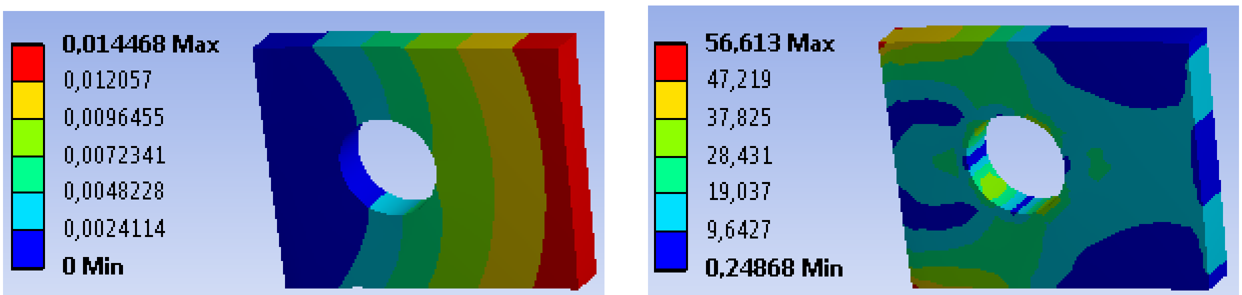
D.2 Lansarea modulului de rezolvare a modelului / Launching the solving module

↵ Solution (A6) → ↵ Solve .

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement and equivalent stresses fields

, Outline; ↵ Solution (A6) → ↵ Total Deformation (fig. a); ↵ → ↵ Show Undeformed WireFrame (fig. a); ↵ Equivalent Stress (fig. b) Graph; ↵ Animation (view animation).



a. *b.*

F. PREPROCESSING OF THE OPTIMIZATION MODEL

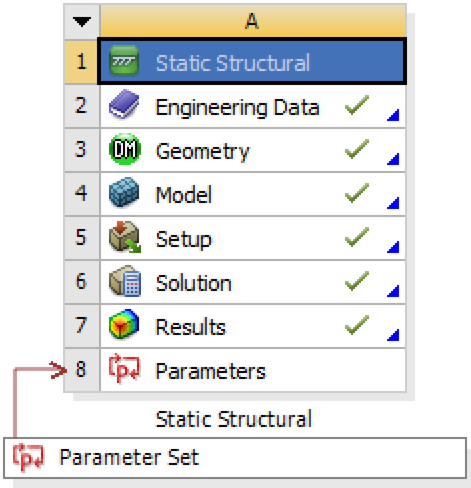
F.1 Setting input (design) and output (status) parameters

Setting input (design) parameters

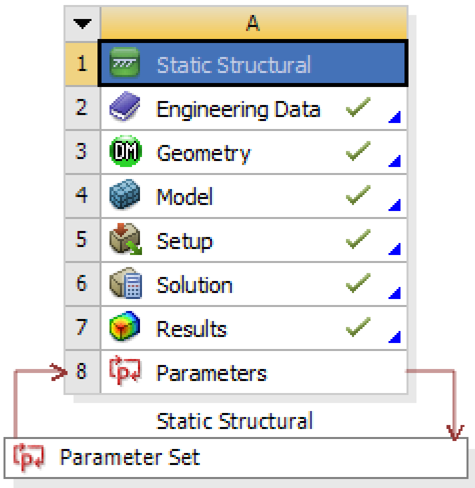
DM: ↲ Sketch1 → Details View, ⊟ Dimensions: 4: [activated with ↲ the button associated with the circle diameter, ↲ D D1] → A: Static Structural - DesignModeler, Parameter Name:, [input the name, *Diameter*], ↲ OK; [activated with ↲ the button associated with the rectangle width dimension, D L2] → A: Static Structural - DesignModeler, Parameter Name:, [input the name, *Width*], ↲ OK (↲ WB → Project Schematic : (the input parameter setting loop appears automatically, fig. a).

Setting output (status) parameters

M, Outline: → ↲ Geometry → ↲ Details of "Geometry", ⊞ Properties, [is activated with ↲ the button associated with the mass, P Mass]; ↲ Solution (A6) → ↲ Equivalent Stress → Details of "Equivalent Stress", ⊟ Results, [is activated with ↲ the button associated with the maximum equivalent voltage, P Maximum], ↲ OK (↲ WB → Project Schematic: (the output parameter setting loop appears automatically, fig. b).



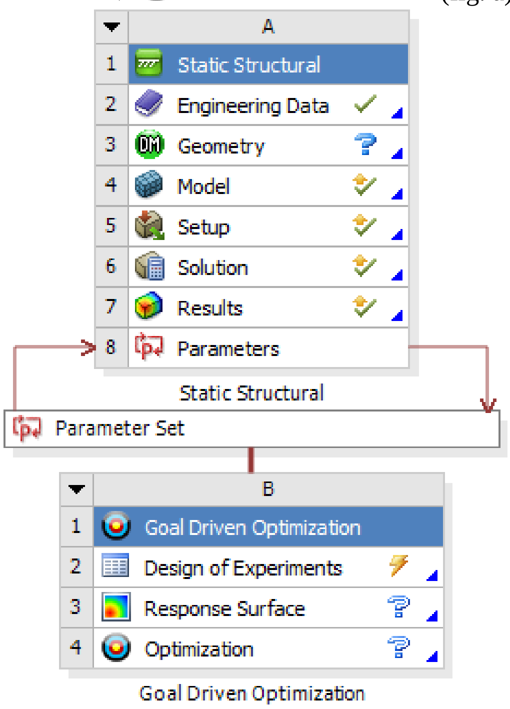
a.



b.

F.2 Launching the optimization module

↲ WB → ↲ ⊞ Design Exploration → ↲ Goal Driven Optimization (fig. a).



a.

F.3 Generating and visualizing feasible solutions

Generating feasible solutions

↵ Design of Experiments → Outline of Schematic B2: Design of Experiments: (↵ View → ↵ Outline, ↵ Properties, ↵ Table, ↵ Chart), ↵ P1 - Diametrul → Properties of Outline A5: P1: Lower Bound, [input the lower value, 14,4], Upper Bound [input the upper value, 17,6]; ↵ P2 - Latimea Properties of Outline A6: P2: Lower Bound, [input the lower value, 36], Upper Bound [input the upper value, 44]. ↵ Update. ANSYS Workbench → ↵ Yes (after processing the table with test variants appears, Table of Schematic B2). ↵ Return to Project

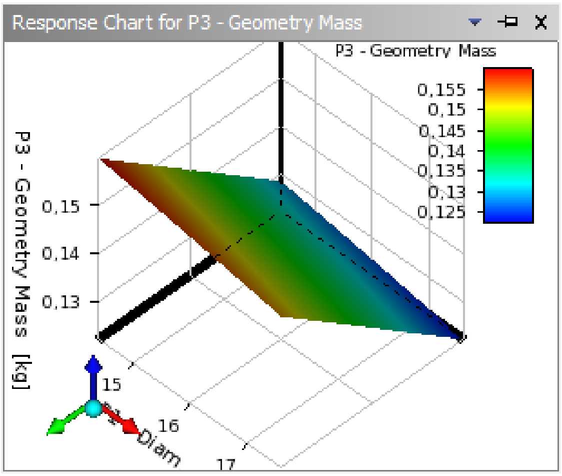
Visualization of feasible solutions

↵ Response Surface → Outline of Schematic B3: Response Surface, ↵ Response Surface → ↵ Update. Outline of Schematic B3: Response Surface → ↵ Response, Properties of Outline A16: Response, Mode, [select from list ↵, ↵3D]; Axes, X Axis, [select from list ↵, ↵P1 - Diametrul]; Y Axis, [select from list ↵, ↵P2 - Latimea]; Z Axis, [select from list ↵, ↵P3 - Geometry Mass] (appear the graph in fig.a); Z Axis, [select from list ↵, ↵ P4 - Equivalent Stress Maximum] (appear the graph in fig.b).

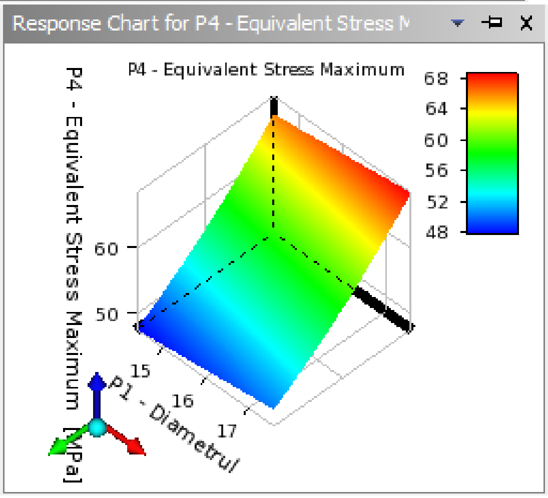
↵ Return to Project.

Table of Schematic B2: Design of Experiments (Central Composite Design : Auto Defined)

	A	B	C	D	E
1	Name	P1 - Diametrul	P2 - Latimea	P3 - Geometry Mass (kg)	P4 - Equivalent Stress Maximum (MPa)
2	1	16	40	0,14122	56,613
3	2	14,4	40	0,14422	55,763
4	3	17,6	40	0,1379	58,223
5	4	16	36	0,12552	67,193
6	5	16	44	0,15692	49,085
7	6	14,4	36	0,12852	65,788
8	7	17,6	36	0,1222	68,589
9	8	14,4	44	0,15992	47,467
10	9	17,6	44	0,1536	50,114



a.



b.

G. SOLVING THE OPTIMIZATION MODEL

↵ → B: ↵ Optimization → Table of Schematic B4: Optimization, Optimization Objectives: Objective, [select in column D from list ↵, ↵Minimize], [select in column E from list ↵,

↵Values <= Target]; Target Value, [input in column D the value limit, 140]. Properties of Outline A2: Optimization, Optimization: Optimization Method, [select from list ↵, ↵NLPQL].

Outline of Schematic B4: Optimization → ↵ Optimization, ↵ Update (appear in window Table of Schematic B4: Optimization lines from fig.a). ↵ Return to Project.

11	Candidate Points				
12	Candidate A	17,6	36	0,1222	68,565
13	Verification A			0,1222	68,589

a.

Obs. The NLPQL (Nonlinear Programming by Lagrangean Quadratic) method is based on the gradient algorithm for models with a single objective function and multiple constraints.

H. POST-PROCESSING OF RESULTS

H.1 Update the original model with the optimal design values

Entering the values of the optimal design parameters

↵ → ↵↵ Parameter Set → Table of Design Points: Current, [input in column B the optimal value, 17,6 (see the table above)], [input in column B the optimal value, 36 (see the table above)]. ↵ Update (the boxes in columns D and E are filled in automatically).

Table of Design Points

	A	B	C	D	E	F	G
1	Name	P1 - Diametrul	P2 - Latimea	P3 - Geometry Mass	P4 - Equivalent Stress Maximum	Exported	Note
2	Units			kg	MPa		
3	Current	17,6	36	0,1222	68,589		

a.

Checking the values of the optimal design parameters

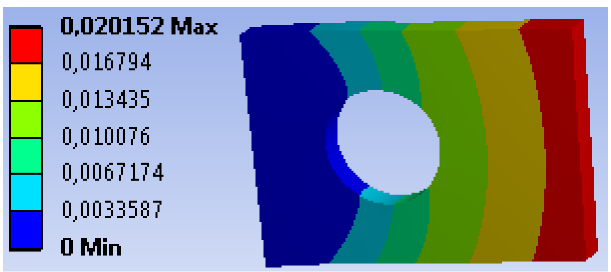
↵ → ↵ Geometry → ↵Sketching (the values of parameters D1 (Diameter) and L2 (Width) updated with the optimal values are observed, 17.6 and 36 respectively).

Upgradare proiect

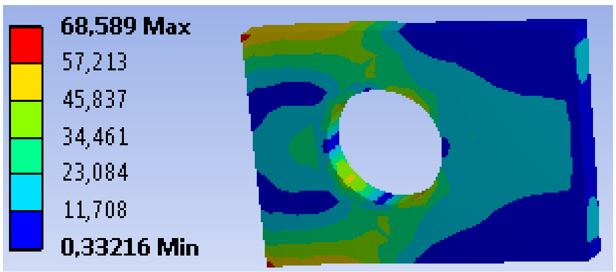
↵ Update Project.

H.2. Visualization of the field of displacements and equivalent post-optimized stresses

, Outline: ↵ ? Solution (A6) → ↵ Total Deformation (fig. a); ↵ → ↵Show Undeformed WireFrame (fig. a); ↵ Equivalent Stress (fig. b) Graph; ↵ Animation (view animation).



a.



b.

I. ANALYSIS OF RESULTS

I.1 Interpretation of results

Following the analysis of the results obtained, as a result of the modeling and post-processing of the results (subchapters E and H), the following are highlighted:

- Following the deformation process of the non-optimized element (D = 16 mm, H = 40 mm) as a result of the action of the force F (subchapter A.2, fig. a) the maximum displacement is observed 0.0144468 mm (subchapter E. 2, Fig. a) in the area of the force action; the maximum equivalent stress has the value 56,614 MPa (subchapter E.2, fig. b) in the embedded area; the mass of the element is 141.22 g (subchapter F.3, Table of Schematic B2).
- Following the deformation process of the optimized element (D = 17.6 mm, H = 36 mm) as a result of the action of the force F (subchapter A.2, fig. a) the maximum displacement is observed 0.020152 mm (subchapter. H.2, Fig. a) in the area of the force action; the maximum equivalent stress has the value 68.589 MPa (subchapter H.2, fig. b) in the embedded area; the mass of the element is 122.2 g (subchapter H.3, fig. a).

I.2 Design studies

The analysis of the above results shows the decrease of the element mass following the finite element solving of the optimization model; at the same time the increase of the maximum displacement (rigidity) is observed.

In order to optimize related to other design restrictions, it is necessary to modify the analysis model, re-adopt the design and status parameters and the objective function. Thus, it is necessary, after the modifications of the analysis and / or optimization model, to solve it by activating the commands ↵ Refresh Geometry ; ↵ Solve . After solving the model, the results are reanalyzed and reinterpreted.

J. CONCLUSIONS

Modeling and analysis with finite elements in this paper were also carried out for didactic purposes following the initiation of the user with the main stages of development of a finite element optimization application in ANSYS Workbench, which emphasizes, above all, the modeling and analysis of a deformable element which is then dimensionally optimized.

The optimization model considered adopted involves the consideration of two geometric parameters as design variables, a state parameter (equivalent voltage) limited below the allowable value and the objective function that involves minimizing the mass of the element.

Following the solution of the finite element model of optimization, adopting the NLPQL method (Nonlinear Programming by Quadratic Lagrangean) which is based on the gradient algorithm for models with a single objective function and multiple constraints, the reduction of the element mass was obtained. maximum (but not exceeding the allowable value) and increasing the rigidity of the element.

Application: FEA-A.11
Compression strained springs

KEY WORDS
Linear Static Analysis, Linear Material, 3D Geometric Model, 3D Finite Element, Linear Finite Element, Classical Verification, Machine Element

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

Many technical products contain mechanical elements that have distinct compact structures, required by the main function to be performed. Representative of this group of components are the elastic elements (springs), the damping elements, the supporting elements (housings), etc. The specificity of these elements, as a rule, is given by their fixed or quasi-fixed connections with the neighboring parts.

The finite element analysis of these components, in order to obtain precise results, presupposes the accurate definition of the solid model, of the restrictions imposed by the connections with the neighboring elements, as well as of the loads.

A.2 Application description

Safety valves are designed to protect tanks, pipes, boilers, boilers or other equipment containing pressurized fluids. These prevent pressure limits from being exceeded when all automatic control and monitoring equipment no longer operates.

Many safety valves (see the figures below, Spring safety valve, Fi-Fi brass body, PN 16, DN ½ ”… 3”, http://www.prestcom-instal.ro, accessed Apr. 2014) have in composition of active elastic elements used to obtain elastic characteristics imposed by the functional requirements. In this case, by changing the coil spring inside the valve, valves with different operating characteristics can be made.

The helical spring has the role of generating an axial force that compensates the force generated by the fluid pressure inside the installation and when the latter increases, the spring will compress by opening the exhaust circuit.

The coil spring used must comply with certain geometrical constraints (to fit within the available space) and to operate (to ensure the force necessary for the operation of the installation, to compress when an overpressure occurs, to generate a sufficiently large stroke so that the section of the circuit is suitable for emergency evacuation and, last but not least, return to working order after restoration of working pressure).

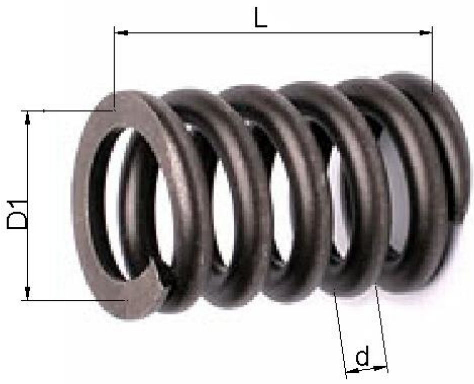
a. *b.* *c.*

A.3 Application goal

In this application it is presented the analysis of the fields of displacements, deformations and tensions in the structure of the elastic element of helical spring type in the valve composition presented above (PN 16, DN 3/4 ”) as well as the values of forces generated by compressing the spring with a certain displacement. which oppose the opening of the valve at nominal working pressures. The values of the geometric and mounting parameters of the helical spring are: d = 2 mm, D1 = 17 mm, the number of turns n = 5 and the pitch t = 5.75 mm. The coil spring is made of spring steel, 50VCr11A, treated at 50-55 HRC.

Axial compression of the spring (3) with the screw (6) in the drawing above will generate a force that compensates for the pressure inside the container on the front surface of the valve piston (according to the product data sheet, the valve piston surface is 283 mm2).

This application monitors the value of the dependence between the value of the compression of the spring and the force generated on the valve piston, in order to design the valve as well as the study of internal stresses in the spring to check if the material meets the operating requirements.



B. THE FEA MODEL

B.1 The model definition

In order to draw up the finite element analysis model associated with the above application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,

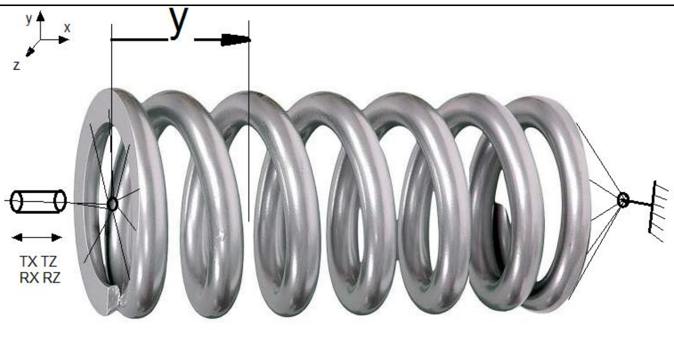
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The geometric shape and dimensions of the helical spring are shown in the adjacent figure. For the analysis, the structure of the helical spring is modeled with 3D finite elements.

In order to simulate the behavior of the helical spring as close as possible to reality, taking into account the increased rigidity of the surfaces on which the spring is placed, two associated rigid elements are introduced.

In order for the analysis model to have the same behavior as the real model, it is necessary to associate boundary conditions that involve translation constraints according to the X and Z directions of the XYZ coordinate system, respectively only motion will be allowed on OY, simulating the placement of the helical arc in the valve seat. In order to generate the translational movement along the OY axis, a rotational translation coupling is introduced associated with the master point of the rigid element at the bottom, corresponding to the point of application of the force.



B.3 Characteristics of the material

For finite element analysis, the strength characteristics of the 50VCr11A spring steel material treated at 50-55 HRC are:

- modulus of longitudinal elasticity, E = 209,000 N / mm2;
- transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Creating of the project

Toolbox: ↵⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural in *Spring*].

Problem type setting (3D)

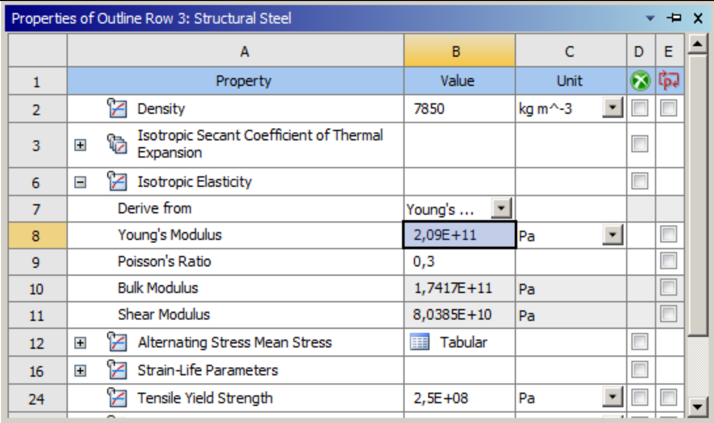
A: ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from the list ↵▼, ↵3D] → [close the window, ↵✕].

Saving of the project

↵ Save As... → Save As, File name: [input name, Spring] → ↵ Save

C.2 Modelling of material and environment characteristics

Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, Young's Modulus, [select from column C (Unit) cu ↵▼, ↵MPa], [input in box from column B (Unit) valoarea / value, 209000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).



Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2,09E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,7417E+11	Pa		
11	Shear Modulus	8,0385E+10	Pa		
12	Alternating Stress Mean Stress	Tabular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

C.3 Geometric modelling

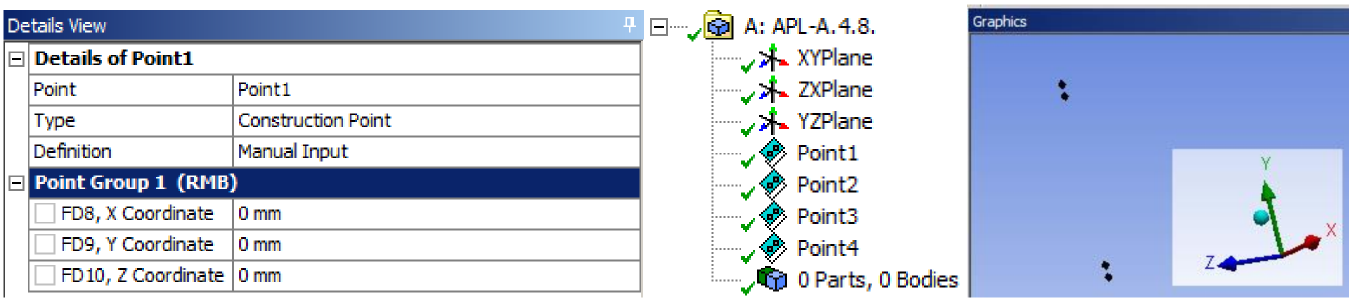
C.3.1 Model loading, DesignModeler (DM)

WB, Project Schematic: ↳ Geometry → ↵DM New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.

C3.2 Spring helix generating

DM, Tree Outline: ↵Sketching → (Look at face/Plane/Sketching) [automatically view of default plane XY Plane]; DM, Modeling → <u>Create</u> → Point [in the 3D modeling area the point P1 is created based on the Cartesian coordinates] → Details View → **Details of Point 1** → Definition: Manual Input; **Point Group 1 (RMB)** → x = 0; y = 0; z = 0 → Generate.

The points P2… P4 are constructed in the same way, using the resulting Cartesian coordinates based on the dimensions given in the model for analysis: P2 (0; 2; 0); P3 (0; 30.75; 0); P4 (0; 32.75; 0).

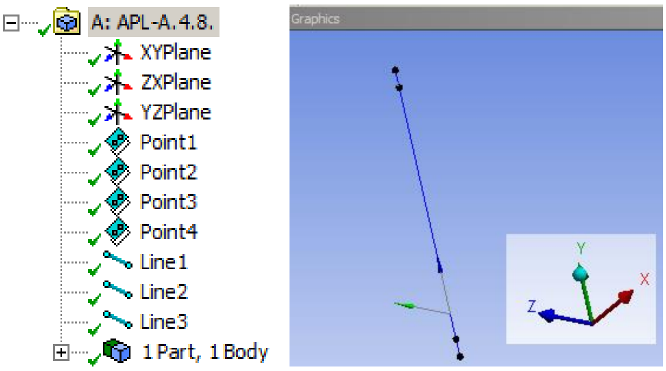


Details View

Details of Point1	
Point	Point1
Type	Construction Point
Definition	Manual Input
Point Group 1 (RMB)	
FD8, X Coordinate	0 mm
FD9, Y Coordinate	0 mm
FD10, Z Coordinate	0 mm

DM → Modeling → <u>Concept</u> → Lines From Points → Details View → **Details of line 1** → Point segments → (points P1 and P2 are selected by holding down the *Ctrl* key) → Apply → Generate.

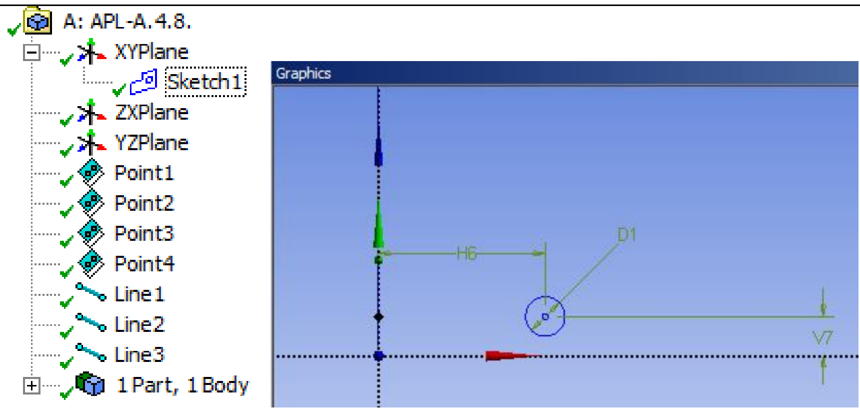
The segments P2P3 and P3P4 are built in the same way.



C3.3 Solid generating

<u>Generating the spring section</u>

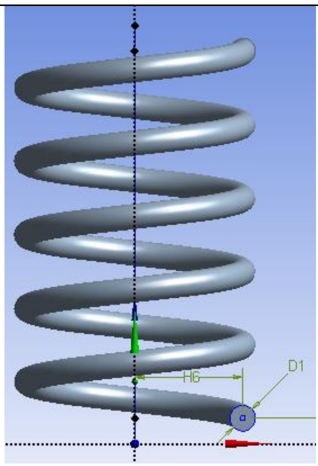
DM → Sketching → <u>Draw</u> → Circle → <u>Dimensions</u> → Diameter → Details View → **Dimensions:1** → D1 = 2 mm → <u>Dimensions</u> → Horizontal → **Dimensions:2** → H6 = 8,5 mm → Vertical → **Dimensions:3** → V7 = 2 mm → Generate.



Generating the central area of the resort

DM → Sweep → Details View → **Details of Sweep1** → Profile: Sketch1 (the previously drawn circle is selected) → Apply → Path: Line2 (select the P2P3 segment from the graphic area) → Apply → Twist Specification ▼: Pitch → Pitch = 5,75 mm → Generate.

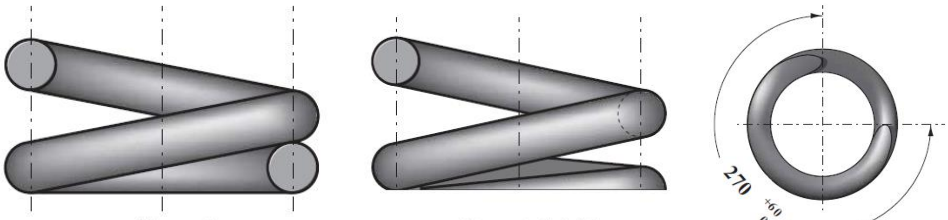
Obs. The geometric characteristics of the spring can also be entered using the number of steps: Twist Specification ▼: Turns → Turns=5 → Generate.



Generating the end areas of the spring

Because in this geometric shape of the spring the contact with other fixing elements will be made in the form of very small seating surfaces (geometrically, the contact is only a point), it is necessary to create appropriate seating areas.

There are two constructive forms, shown in the figures below. For this application, form 2 will be used. To make the polished ends, the spring must be extended on one side and on the other with a coil with a smaller step, so that a contact spot of at least 270° can be made when milling.



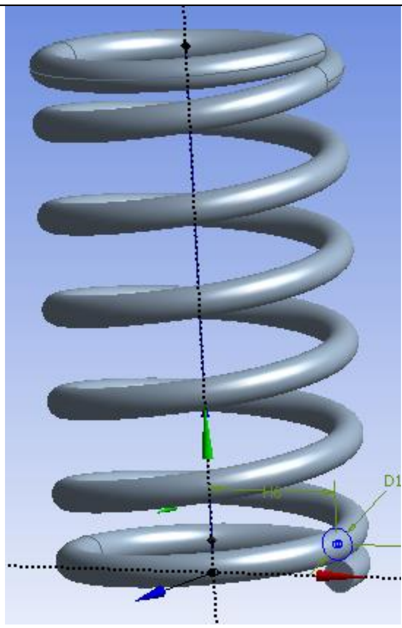
capăt aşezat

capăt aşezat şi şlefuit

DM → Sweep → Details View → **Details of Sweep2** → Profile: Sketch1 (select the circle previously drawn, located at the axis on the axis Oy = 2 mm) → Apply → Path: Line1 (the P1P2 segment is selected from the graphics area) → Apply → Twist Specification ▼: Turns → Turns = 1 → Generate.

DM → Sweep → Details View → **Details of Sweep3** → Profile: (select the circular section of the spring located at 32.75 mm on the Oy axis) → Apply → Path: Line3 (select the P3P4 segment from the graphics area) → Apply → Twist Specification ▼: Turns → Turns = 1 → Generate.

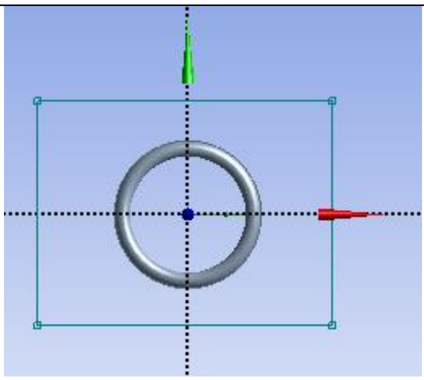
The milling surfaces of the spring are milled by extruding command (using the option to cut from the material) some rectangular surfaces drawn in planes parallel to the xOz plane at elevations of 1.25 mm and 31.5 mm, respectively.

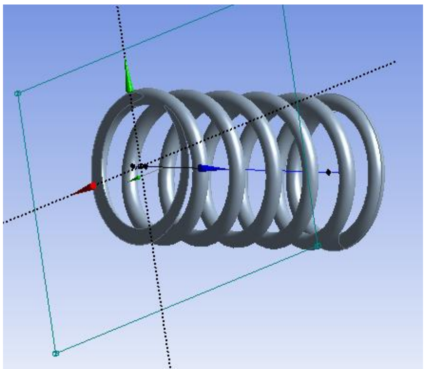


DM → New Plane → Details View → **Details of Plane4** → Type: From Plane → Base Plane: ZXPlane → Transform 1 (RMB): Offset Z → Value 1: 1,25 mm → Generate. Repeat the sequence of commands to generate the plan at 31.5 mm.

DM → Plane4 → → Sketching → Draw → Rectangle (a rectangle is constructed so that the spring fits into it) → Modeling → Extrude → → Details View → **Details of Extrude3** → Geometry: Sketch3 (select the rectangle from the Plane4 plane) → Operation : Cut Material → Direction: Reversed → Depth: 10 mm → Generate.

Follow the same steps for the other end of the spring.





C.3.4 Save of geometric model

DM, ↵ (Save Project) → File → Close Design Modeler.

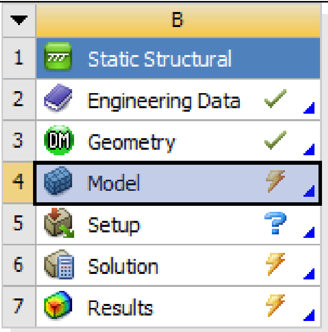
C.4 Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

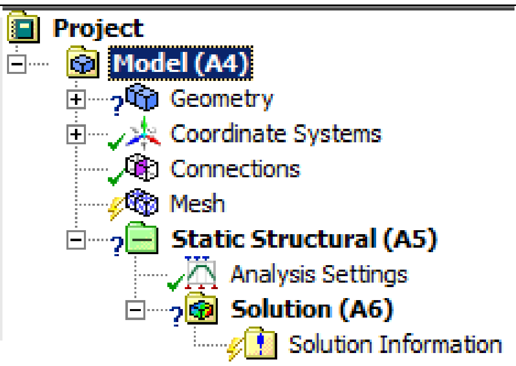
Launching of the finite element modeling module

WB, Project Schematic: Model → ↵ Edit... → [launch module *Mechanical [ANSYS Multiphysics*].

↵ Geometry → Details of "Geometry" → **Definition** → Element Control : Program Controlled.



AEF-A.4.6.



Setting the unit of measure system

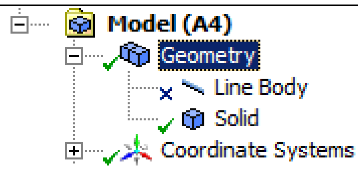
: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

C.4.2 Model discretization and analysis type setting

Disable help items

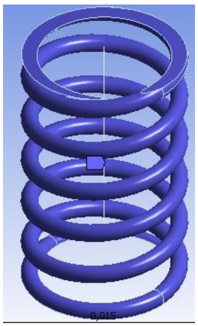
P1P2, P2P3 and P3P4 segments will be disabled so as not to affect discretization and other subsequent operations.

, ↵ Geometry → ↳ Line Body → Suppress Body.



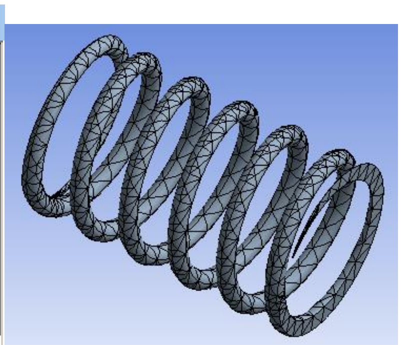
, ↵ Mesh → ↵ Mesh Control → ↵ Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ spring geometry, using the selection filter (Body)] Apply; **Definition Element** → Size: 0,002 m → Update.

For a proper view of the discretization, this will be done: ↵ Show Mesh.



Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	2,e-003 m
Behavior	Soft



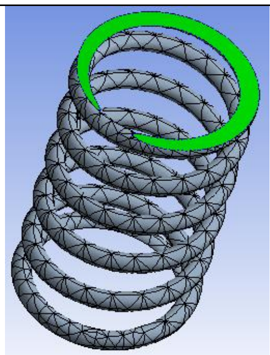
C.5 *Supports and restraints modelling*

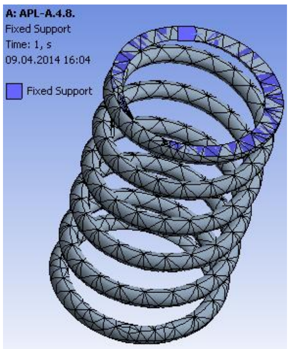
Input the gravitational acceleration

Because the weight of the spring is very small (approximately 75 g), the influence of the weight force (0.73 N) on the results of the analysis is very small, taking into account the value of the other stresses.

Input restraint

, ↵ Static Structural (A5) → Supports → Fixed Support → Details of „Fixed Suports" → **Scope** → Geometry: [select with ↵ the milled surface at the end of the spring at a height of 1.25 mm, using the (Face)] → Apply.

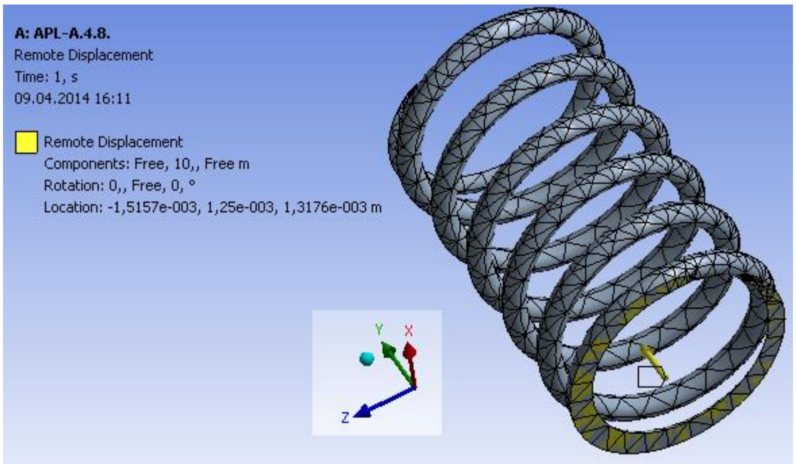




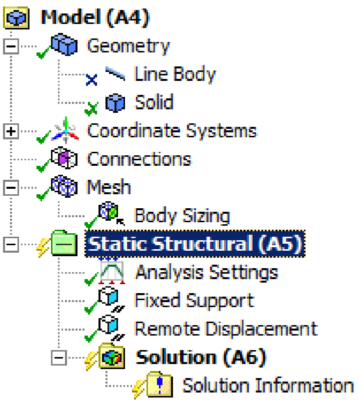
C.6 Load modeling
Input Remote Displacement

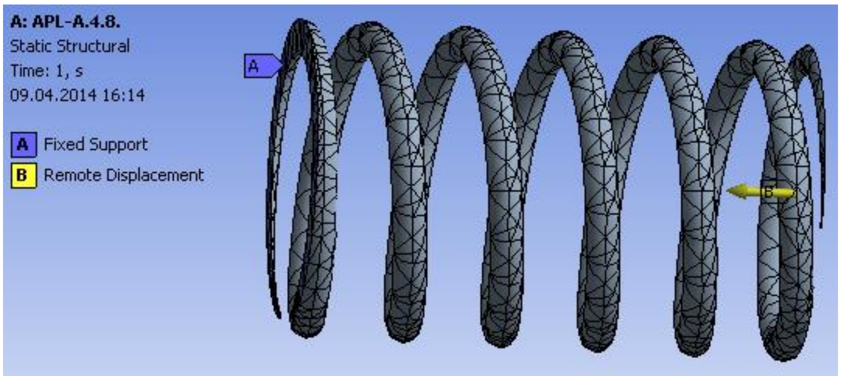
, ↲ Static Structural (A5) → Supports → Remote Displacement → → Details of „Remote Displacement” → **Scope** → Geometry: [select with ↲ the milled surface at the end of the spring at a height of 31.5 mm, using the (Face)] → Apply; **Definition** → X Component: Free, Y Component: 10 mm, Z Component: Free; Rotation X: 0, Rotation Y: Free, Rotation Z: 0, Behavior: Rigid.

A virtual rigid element was formed which consists of the front milled surface of the spring, located at an elevation of 31.5 mm, which is connected to a rotary translation coupling on the OY axis and which is printed a displacement of 10 (15) mm in the direction OY.



The constraints and loads of the resort will look like the figure below.





D. SOLVING THE FEA MODEL

D.1 Setting the convergence criterion for solving the model

, Outline: → ↲ Solution (A6) → ↲ Solution Information, Details of "Solution Information",

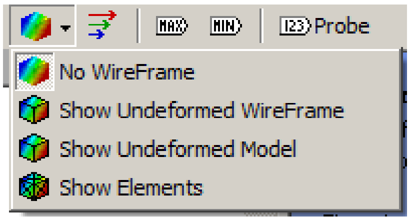
↲ Solution Information: ↲Solution Output → [selecting from the list with ↲ , ↲Force Convergence] (the force convergence criterion is adopted). These steps will be repeated and chosen „Displacements Convergence”.

D.2. Setting results
Selecting the types of results

In order to select the final data types to be analyzed after the launch of the calculation module, follow the series of commands presented below.

Outline: ↳ Solution (A6) → ↵ Insert → ↵Deformation → ↵ Total. [use the commands in the open command box with ↳]. The same result can be obtained by using commands: ↳ Solution (A6) → ↵ Insert → Deformation → Total [the buttons in the menu bars are used] and: ↳ Solution (A6) → ↵ Insert → Deformation → Directional. ↳ Solution (A6) → ↵ Insert → Stress → Equivalent (von-Mises). ↳ Solution (A6) → ↵ Insert → Stress → Error. ↳ Solution (A6) → ↵ Insert → Stress → Maximum Principal. ↳ Solution (A6) → ↵ Insert → Strain → Equivalent (von-Mises). ↳ Solution (A6) → ↵ Insert → Strain → Vector Principal. ↳ Solution (A6) → ↵ Insert → Tools → Stress Tool → Safety Factor. ↳ Solution (A6) → ↵ Insert → Energy → Strain Energy. Next, the other types of results to be analyzed are set, respectively the reactions in the supports: ↳ Solution (A6) → ↵ Insert → Probe → Force Reaction; ↳ Solution (A6) → ↵ Insert → Probe → Moment Reaction.
D.3 Launching the solving module
Outline: Analysis Settings → Details of "Analysis Settings" → **Solver Controls** → Large Deflection : On → ↵ Solution (A6) → Solve.

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field
For suggestive results, set the view scale of the menu bars: Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale). *Total deformation viewing* Outline: ↵ Solution (A6) → ↵ Total Deformation → Tab-ul Graph → Animation. If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto). Various forms of distorted state representation can be used by calling the (Edge) button. *Show Underformed WireFrame* will be selected, an option that displays the undeformed and warped models in the same representation.  The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation 2 Sec (Auto). At the same time, the result can be saved as a video file using the *Export Video File* command.

Visualization of the deformation in one direction

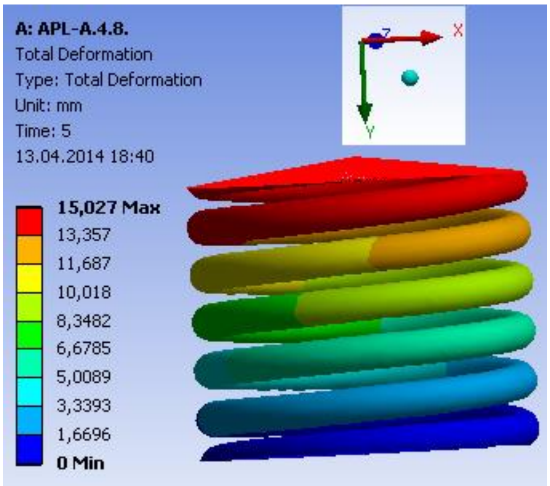
Outline: ↲ Solution (A6) → Directional Deformation → Graph → Animation.

If you want to view it in another direction, follow the steps below:

Outline: ↲ Solution (A6) → Directional Deformation → Details of „Directional Deformation" → **Definition** → Orientation : X Axis → Solve.

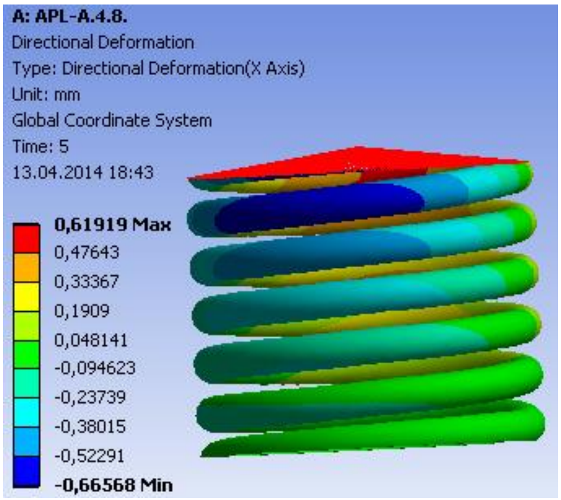
Total Deformations

↲ Solution (A6) → Total Deformation → Graph → Animation.



Directional Deformation

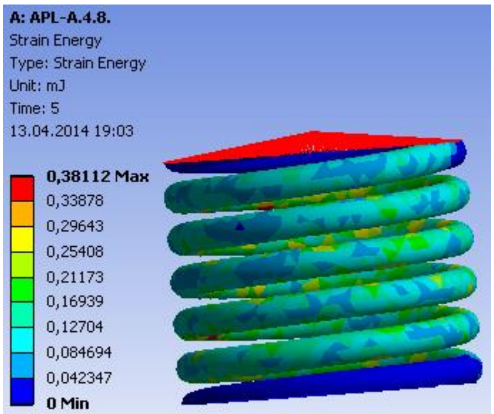
↲ Solution (A6) → Directional Deformation → Graph → Animation.



E.2 Visualizing the fields of stresses, forces and moments

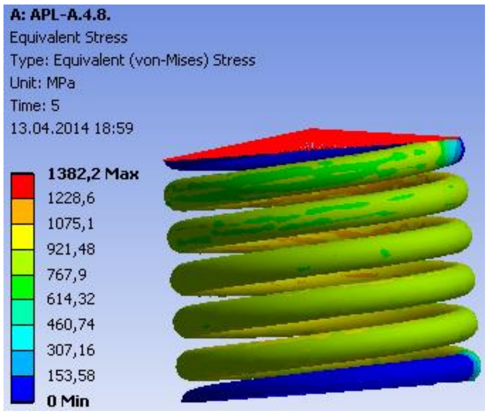
Strain Energy

↲ Solution (A6) → Strain Energy → Graph → Animation.



Equivelent Stress

↲ Solution (A6) → Equivalent Stress → Graph → Animation.

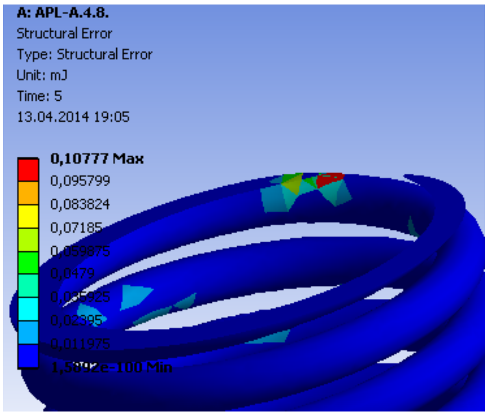


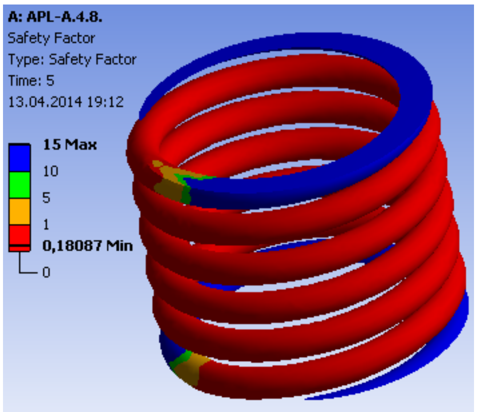
Structural Error

↲ Solution (A6) → Structural Error → Graph → Animation (sau Tabular Data).

Safety Factor

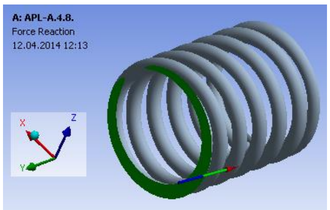
↲ Solution (A6) → Safety Factor → Graph → Animation (sau Tabular Data).





Force reaction in support

↵ **Solution (A6)** → Force Reaction → Graph → Tabular Data.

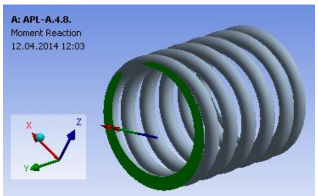


Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	1,75	-2,2517e-002	-34,33	-3,3442	34,493
2	2,25	-4,279e-002	-44,059	-4,3879	44,277
3	2,75	-7,1138e-002	-53,753	-5,4735	54,031
4	3,5	-0,13174	-68,226	-7,1793	68,603
5	4,	-0,18594	-77,828	-8,3664	78,277
6	4,5	-0,25271	-87,392	-9,5883	87,916
7	5,	-0,33295	-96,916	-10,839	97,521

Moment reaction in support

↵ **Solution (A6)** → Moment Reaction → Graph → Tabular Data.



Tabular Data

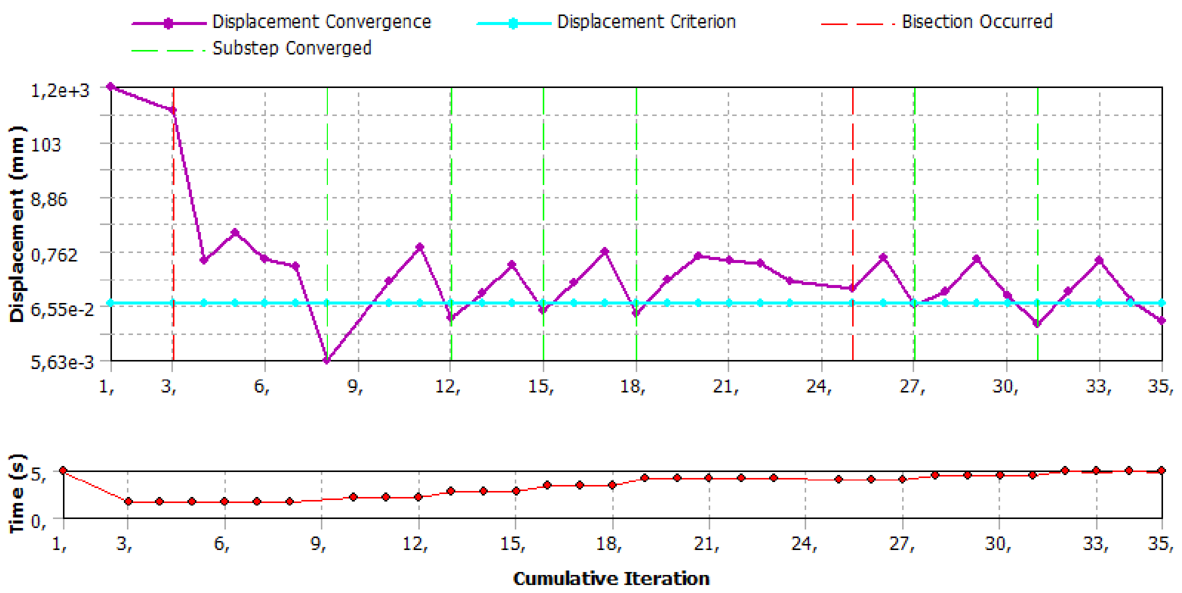
	Time [s]	Moment Reaction (X) [N·mm]	Moment Reaction (Y) [N·mm]	Moment Reaction (Z) [N·mm]	Moment Reaction (Total) [N·mm]
1	1,75	87,054	5,0396	-49,646	100,34
2	2,25	109,64	6,5898	-63,599	126,93
3	2,75	131,09	8,2072	-77,444	152,48
4	3,5	160,9	10,713	-97,993	188,7
5	4,	179,04	12,445	-111,52	211,3
6	4,5	195,67	14,194	-124,9	232,57
7	5,	210,64	15,98	-138,09	252,38

E.3 Visualizing the convergence of solutions

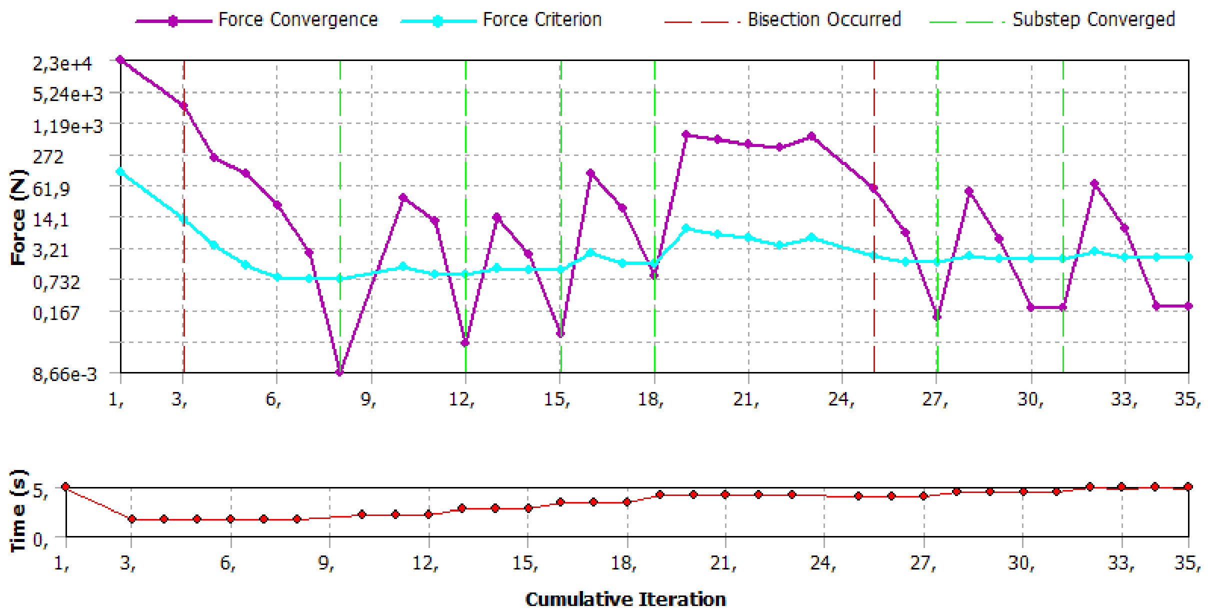
Outline: ↵ **Solution (A6)** → Solution Information → Details of "Solution Information" → **Solution Informations** → Solution Output: Displacements Convergence.

Repeat these steps and select "*Force Convergence*".

Displacement Convergence

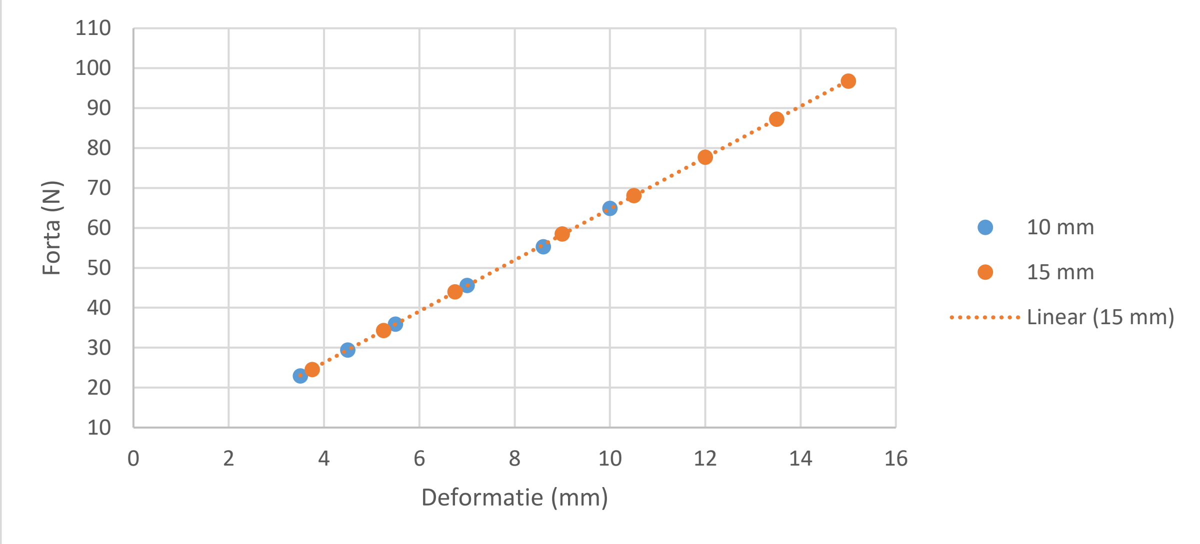


Force Convergence



F. ANALYSIS OF RESULTS

F.1 Interpretation of results
The characteristic of the spring was drawn after performing two simulations with deformations of 10 mm (blue dots), respectively 15 mm (red dots). A perfect overlap of the results is observed, and it can be concluded that this graph is correct.



From the point of view of the deformations in the area 0… 15 mm, it is observed that the graph is a straight line segment, so in this interval the spring works in the elastic zone of the deformations. The value of the force generated by the spring can be extracted from the graph, depending on the value of its deformation.

For example, at a spring compression of 10 mm, the force generated on the valve piston is about 65 N. According to the technical data of the safety valve analyzed, the valve seat has a front surface of 283 mm^2. According to the relation p = F / S, the value of the nominal pressure obtained by the valve is obtained: p = 2.3 bar. For a spring compression of 15 mm, the operating pressure becomes p = 3.4 bar.

Because the maximum compression of this spring depends on the pitch, the number of turns, and the diameter of the turn:

$x = (p - d) \cdot n = (5{,}75 - 2) \cdot 5 = 18{,}75\ mm$,

it can be concluded that this valve will operate in a range of working pressures between 0.7 barr (corresponding to a 2 mm spring compression) and 3.75 barr (for 16 mm compression).

F.2 Accuracy and convergence analysis

The information regarding the deformations, corroborated with the information regarding the equivalent stresses, the structural error, the convergence of the solutions lead to the conclusion that the spring withstands the loads without problems, the values of the maximum stresses not exceeding the allowed limit value of the material. Increased attention must be paid to the connections at the exit of the spring propeller, at both ends, these two areas being important concentrators of stresses and a discretization finish is required, here appearing the maximum structural errors. The much lower values of the structural error field (max 0.107 mJ, subchapter E.2) indicate that the stress values are close to the exact ones. In addition, from subchapter. E.3 highlights the fast convergence (25 steps) of the model solving algorithm and the calculation time is reduced.

G. CONCLUZII / CONCLUSIONS

In order to use the valve for higher ranges of working pressures (its body withstanding pressures of 16 barr), it is necessary to change the spring with some with different characteristics: either with a larger coil diameter or better materials.

To demonstrate the concept, the diameter of the coil will be changed from 2 mm to 2.5 mm, as follows:

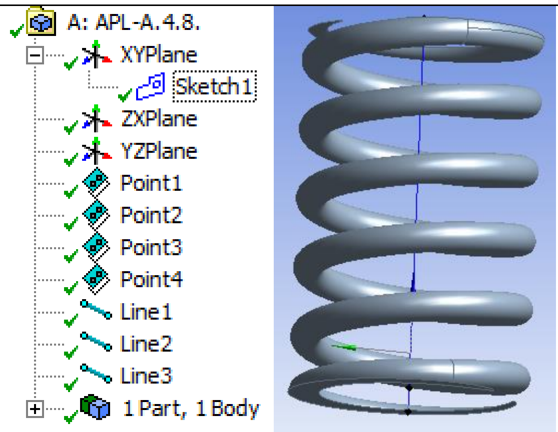
→ Sketching → ↵ Sketch1 → Details View →

Dimensions:3 → D1 = 2,5 mm → Generate.

→ Sketching → ↵ Sweep2 → Details View →

Details of Sweep2 → Twist Specifications : Pitch →

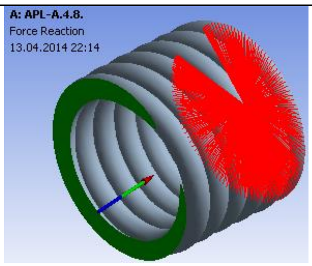
Pitch = 2,5 mm Generate.



Repeat the steps for Sweep3.The spring in the figure from above will be obtained. Next, the analysis operations will be performed according to the steps presented above.

→ File → Refresh All Data → Solve.

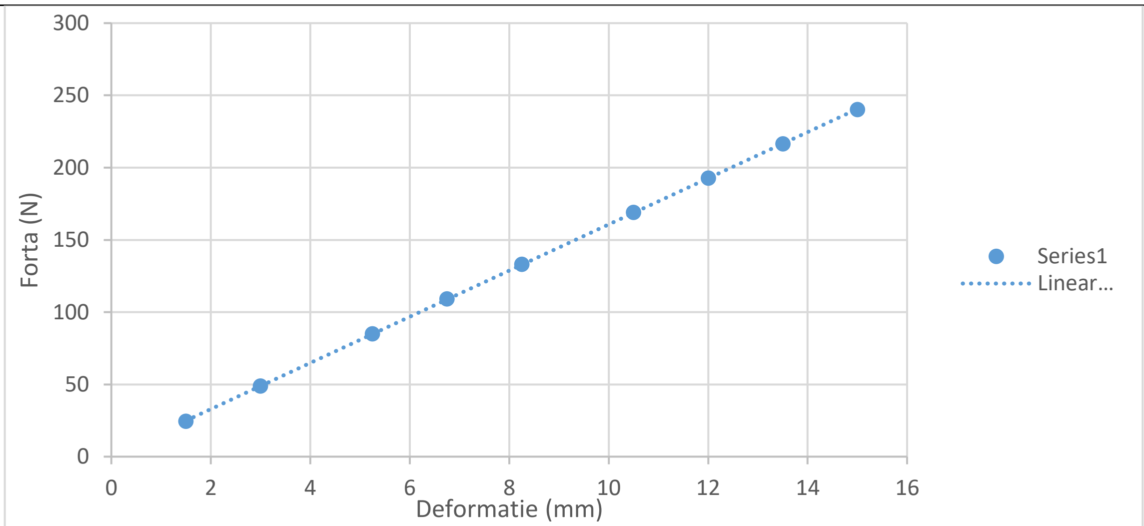
↵ **Solution (A6)** → Force Reaction → Graph → Tabular Data.



Tabular Data

	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total) [N]
1	0,5	-4,4687e-003	-24,4	-2,2287	24,501
2	1,	-1,9466e-002	-48,712	-4,5406	48,923
3	1,75	-6,717e-002	-85,019	-8,1745	85,411
4	2,25	-0,11957	-109,11	-10,713	109,64
5	2,75	-0,19182	-133,12	-13,346	133,79
6	3,5	-0,3436	-168,97	-17,471	169,87
7	4,	-0,47781	-192,76	-20,33	193,83
8	4,5	-0,6425	-216,45	-23,265	217,7
9	5,	-0,8382	-240,05	-26,259	241,49

Based on the data obtained by simulating a 15 mm compression, the characteristic of the 2.5 mm diameter coil spring is obtained. The maximum deformation of this spring is 16.25 mm. Depending on the characteristics of this spring, the nominal working pressures will be in the range (1,2; 8,4) barr. It can be seen that for any spring, following a fairly simple analysis, it can be checked whether it will work properly but the values of the nominal working pressures of the safety valves can also be estimated.



Application: FEA-A.12
Torsional springs

KEY WORDS

Linear Static Analysis, Linear Material, 1D Geometric Model, 1D Finite Element, Linear Finite Element, CATIA Geometric Modeling, Classical Method Verification, Machine Element

CONTENT

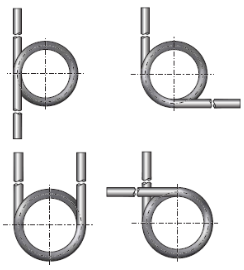
B. PROBLEM DESCRIPTION

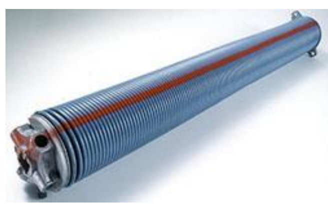
A.1 Introduction

There are mechanical components in many technical products that have distinct compact structures required by the main function to be performed. Representative of this group of components are the elastic elements (springs), the damping elements, the supporting elements (housings), etc. The specificity of these elements, as a rule, is given by their fixed or quasi-fixed connections with the neighboring parts. The finite element analysis of these components, in order to obtain precise results, presupposes the accurate definition of the solid model, of the restrictions imposed by the connections with the neighboring elements, as well as of the loads.

A.2 Application description

Springs are machine elements that, due to the shape and elastic properties of the materials, store the mechanical work of external forces, at deformation, and return it, almost in whole or in part, in the period of return to the original shape. These cylindrical coil springs are made of round diameter round wire. The introduction of force or torque occurs through the arm at the beginning and end of each spring.
These springs have a linear torque characteristic and can be made by cold or hot forming. There may be various constructive forms, shown in the figure below. They can be used in various applications, some of which are shown in the images below.



A.3 Application goal

In the case of this application, the analysis of the fields of displacements, deformations and tensions of an elastic element of curved bar type from the composition of the devices presented above is presented. The values of the geometric parameters of the spring were taken from the specialized literature, as follows: d = 2.5 mm - the diameter of the coil; D_m = 50 mm - average diameter; n = 8.5 turns; Δ = 0.5 mm distance between turns.

B. THE FEA MODEL

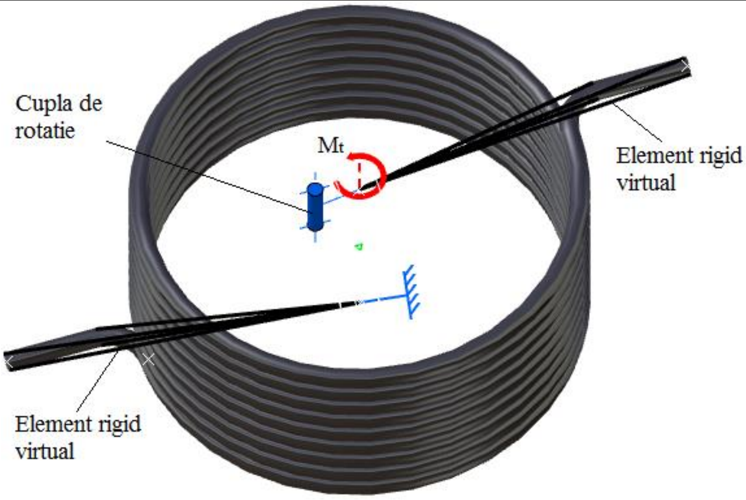
B.1 The model definition

In order to draw up the finite element analysis model associated with the above application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The geometric shape and dimensions of the helical spring are shown in the figure below.



B.3 Characteristics of the material

For finite element analysis, the strength characteristics of the 50VCr11A spring steel material treated at 50-55 HRC are:

- modulus of longitudinal elasticity, E = 209,000 N / mm2;
- transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Creating of the project

WB, Toolbox: ↲⊟ Analysis Systems → ↲↲ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural în / in *Torsional spring*].

Problem type setting (3D)

A : ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from the list ↵▼, ↵3D] → [close the window, ↵✕].

Saving of the project

↵ Save As… → Save As, File name: [input name, *Torsional Spring*] → ↵ Save

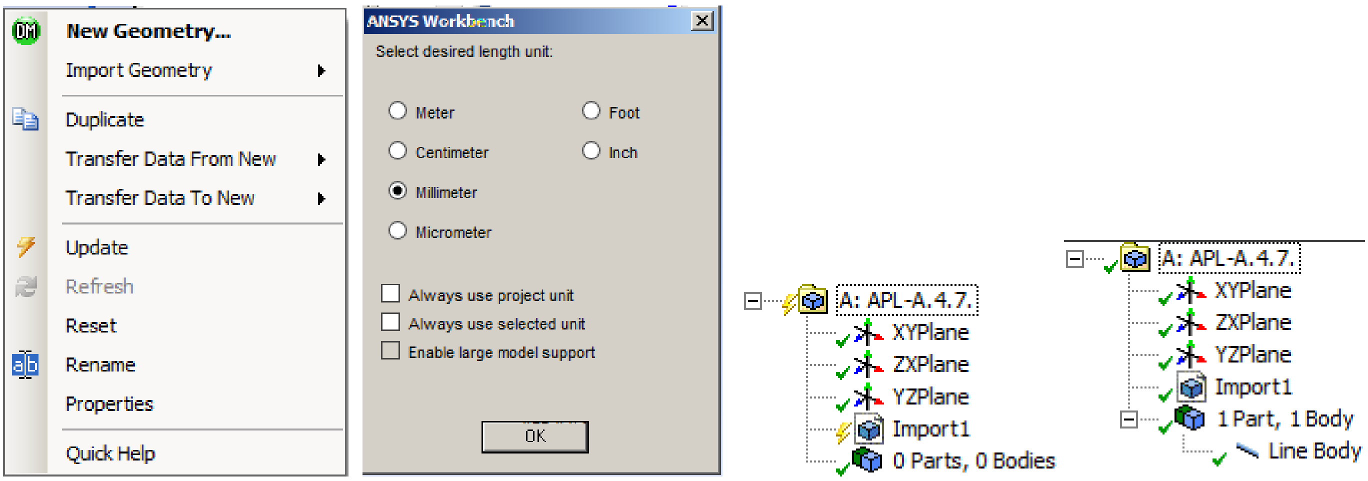
C.2 Geometric modelling

C.2.1 Importing the geometric model of the spring

This application will aim to use a geometric model made in another drawing / design environment. The model is made in advance in CATIA v5R21 in the form of a 1D body, with the geometric construction data presented in the Model section for analysis. The file, originally saved in the specific format of the CATIA software (.catpart) will be saved under the extension of a universal transfer format (.igs or .stp).

Toolbox ↳ Geometry → Import Geometry → Browse → (navigate to the directory structure of the HDD and identify the file T*orsiunal Spring-1D.igs*) → ↵ (OK);

Toolbox ↳↳ Geometry → **ANSYS Workbench**: Select desired length unit: ◉ Millimeter → ↵ (OK) → DM → Generate.



C.2.2 Creating the geometric model in CATIA

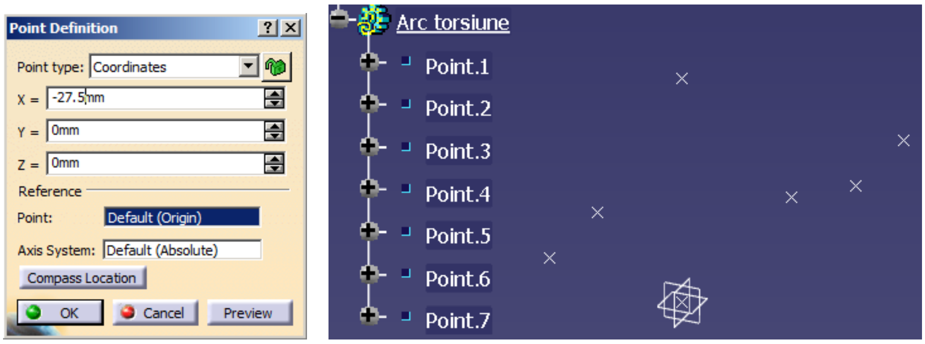
Activating the shape generation module and setting the unit of measure for lengths

CATIA → Start → Shape → Generative shape design → **New part**: New part name: *Spring*.

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

Generating of reference points

(Point) → **Point Definition**: X -27.50mm, Y 0mm, Z 0mm; ↵OK. [similarly, the coordinates of some auxiliary points are introduced - which will help to achieve the geometry of the arc P2(-32.5,0,0), P3(-44.5,5,0), P4 (32.5, 5, 25.5), P5(44.5, 5, 25.5) and the points P6(0, 0, 0) şi P7(0, 0 , 25.5) used to create virtual rigid elements].

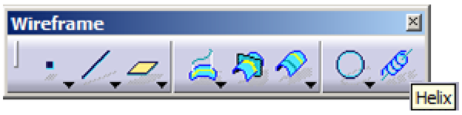


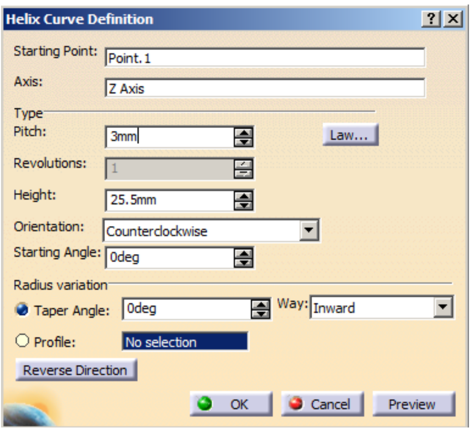
Helical spring generation

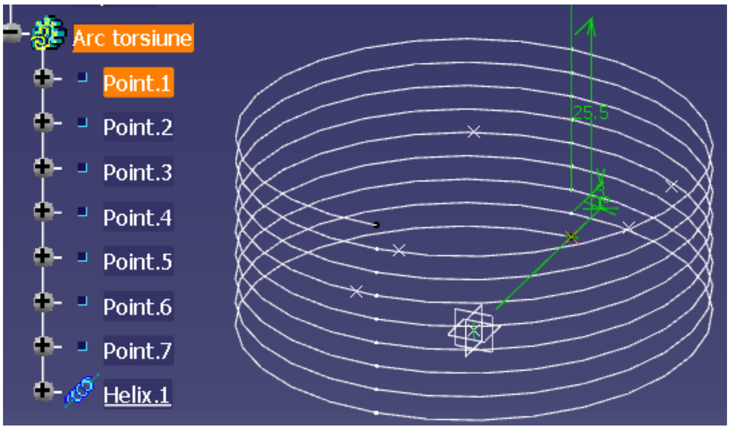
(Helix) → **Helix Curve Definition** select with the mouse in the graphic area or in the tree structure the point Point.1 for Starting Point and for Axis, with the help of a right click on the selection box choose the OZ axis, then fill in the step values: 3 mm and the height of the spring = 8.5 steps x 3 mm = 25.5 mm], ↵ OK.

Obs.

The Helix command can be found in the Wireframe menu.



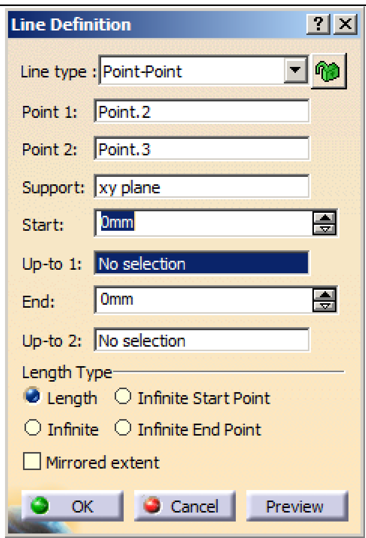


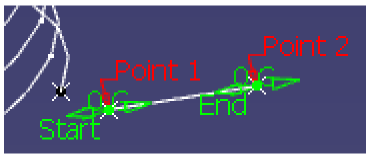


C.2.3 Generating of clamping areas at the ends of the spring

Generating of clamping areas at the ends of the spring

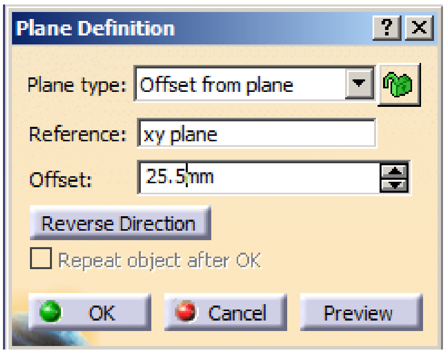
(Line) → [se construieşte un segment de dreaptă din extremitatea arcului de cerc realizat anterior în punctul P3 / a line segment is constructed from the end of the circle arc previously made at point P3] → **Line Definition** → Line type: Point to point, Point 1: Point.2, Point 2: Point.3, Support: xy plane ↵ OK





This command is repeated for the other end of the spring, using points P4 and P5, as follows::

(Plane) → **Plane Definition** → [A plane parallel to XOY is constructed, one at a distance (Offset) of 25.5 mm from XOY] ↵ OK.

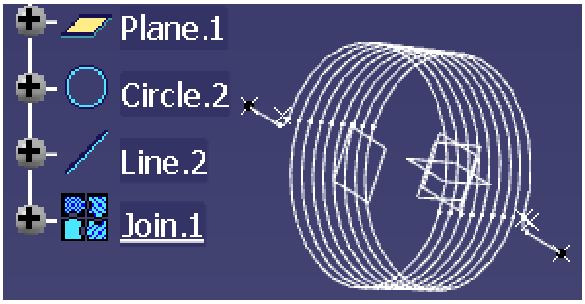


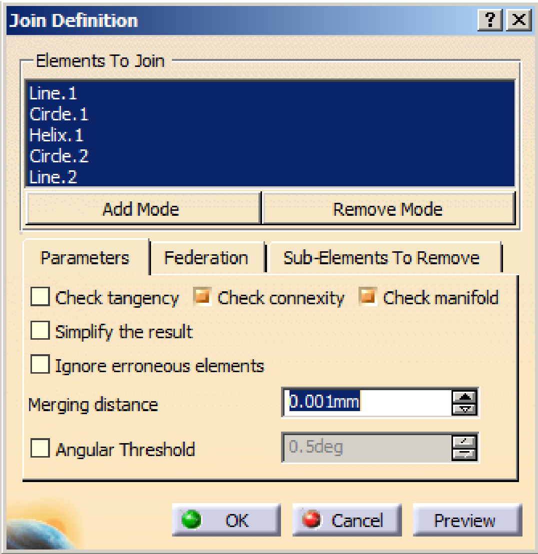
(Line) → [a right segment is constructed from the end of the circle arc previously made at point P5] → **Line Definition** → Line type: Point to point, Point 1: Point.4, Point 2: Point.5, Support: Plane.1 ↵ OK

(Corner) → [the 5 mm pitch helix is connected to the P1P2 and P4P5 segments with a radius of 5 mm] → **Corner Definition** → Circle type: Center and point, Center: Point.2, Point: Point.1, Support: xy plane, Start: 0 deg, End: 90deg ↵ OK

Joining the five segments of the arc

(Join) → **Join Definition** [select Helix.1, Circle.1, Line.1, Circle.2, Line.2] ↵ OK.





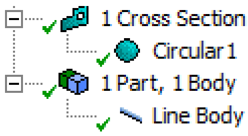
Generating the spring section

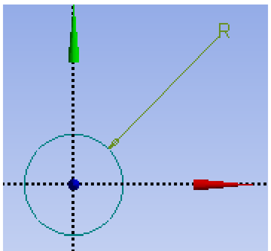
ANSYS Workbench → → Modeling →

Concept → Cross Section → Circular →

Details View → **Dimensions:1** → R1 = 1,25 mm →

Generate.



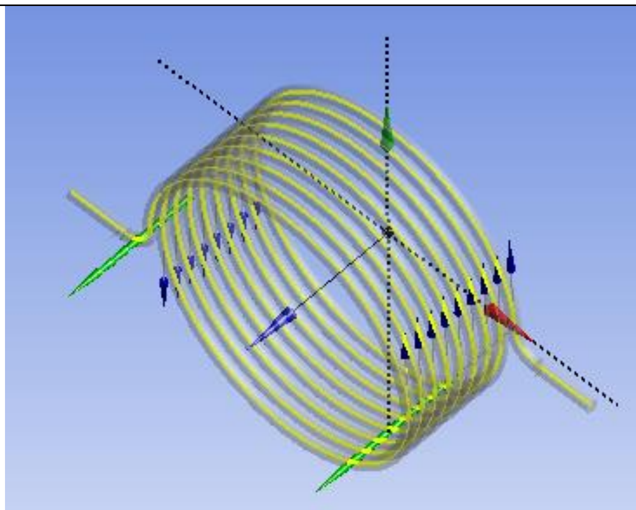


Assigning a cross-sectional profile to the spring

Generating a profile does not mean assigning it to a 1D part. To complete the procedure, proceed as follows

→ ↵ Line Body → Details View → **Details of Line Body** → Cross Section : Circular 1;

Modeling → View → Cross Section Solids [the 3D viewing option is activated] → Generate .



Building a point in the 3D space of the spring

→ Create → Point → Details View → **Details of Point 1** → Definition : Manual Input; **Point Group 1 (RMB)** → X Coordinate = 0, Y Coordinate = 0, Z Coordinate = 0 → Generate.

This point will be used to create a virtual rigid element for the introduction of forces.

C.3 Modelling of material characteristics

Changing the value of the modulus of elasticity

, Project Schematic: ↳ Engineering Data ✓ → ↲ Edit... → Outline of Schematic A2: Engineering Data: ↲ Structural Steel , Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus , Young's Modulus , [select from column C (Unit) cu ↲ , ↲MPa], [input in column B (Unit) valoarea / value, 209000] → ↲ Update Project → ↲ Return to Project (the other parameters remain the default).

Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	⊞ Isotropic Secant Coefficient of Thermal Expansion				
6	⊟ Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2,09E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,7417E+11	Pa		
11	Shear Modulus	8,0385E+10	Pa		
12	⊞ Alternating Stress Mean Stress	Tabular			
16	⊞ Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

C.4 Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

Launching of the finite element modeling module

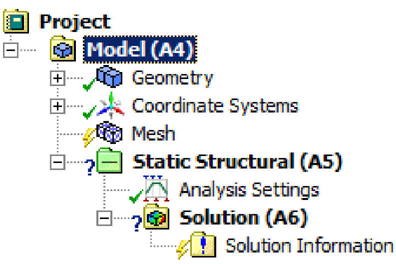
, Project Schematic: ↳ Model → ↲ Edit... → [launching module *Mechanical [ANSYS Multiphysics*].

Setting the unit of measure system

: ↲ Units → ↲ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default). ↲ Geometry → Details of "Geometry" → **Definition** → Element Control : Program Controlled .

	B	
1	Static Structural	
2	Engineering Data	✓
3	Geometry	✓
4	Model	⚡
5	Setup	?
6	Solution	⚡
7	Results	⚡

AEF-A.4.6.



Details of "Geometry"

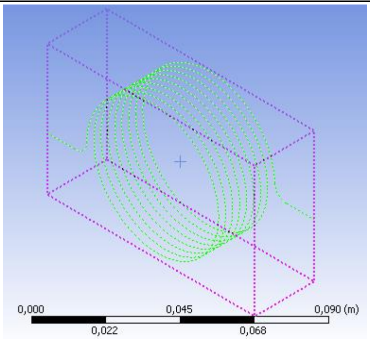
⊟ **Definition**	
Source	C:_Documente\Carte MEF\AEF-A_Aplicatie ferma 1D\F...
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Body Color
⊞ **Bounding Box**	
⊞ **Properties**	
⊞ **Statistics**	
⊞ **Basic Geometry Options**	
⊞ **Advanced Geometry Options**	

Setting the material characteristics

Outline: ↲⊞ Geometry → ↲Line Body → Details of "Line Body , **Material** : ↲Assignment → [select from list↲ , ↲Structural Steel (set by default)].

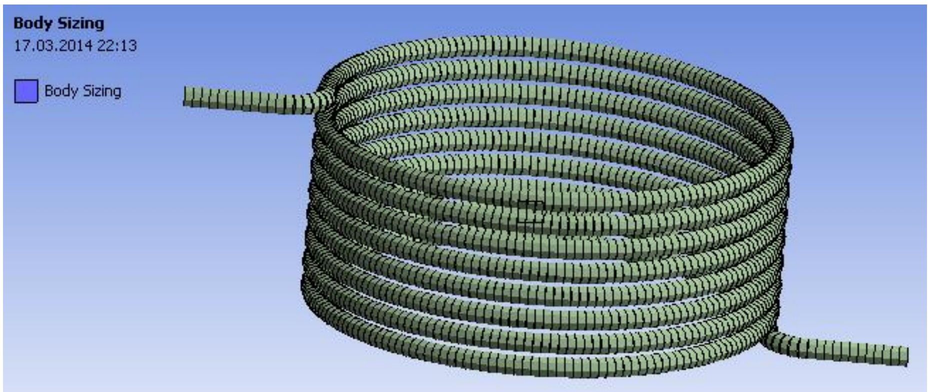
C.4.2 Model discretization and analysis type setting

, Outline: ↲ Mesh → Mesh Control → Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [select with ↲ the geometry of spring using (Body)] Apply; **Definition Element** → Size: 0,001 m → Update



Details of "Body Sizing" - Sizing	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	1,e-003 m
Behavior	Soft

For a proper view of the discretization, this will be done: ↵ Show Mesh.



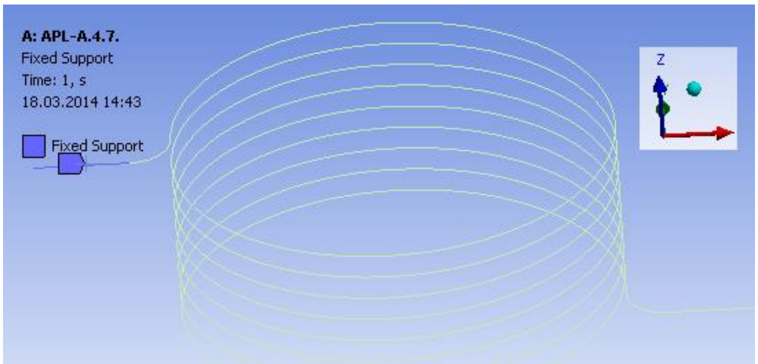
C.5 *Supports and restraints modelling*

Input the gravitational acceleration

Since the weight of the spring is very small (about 58 g), the influence of the weight force (0.56 N) on the analysis results is very small, taking into account the value of the applied torque (which is 1 Nm, which corresponds to a force of of 40 N acting at the helix end point).

Input restraint

Outline: ↵ Static Structural (A5) → Supports → Fixed Support → Details of „Fixed Suports" → **Scope** → Geometry: [select with ↵ the end segment of the spring at a height of 25.5 mm, using the selection filter (Edge)] → Apply.

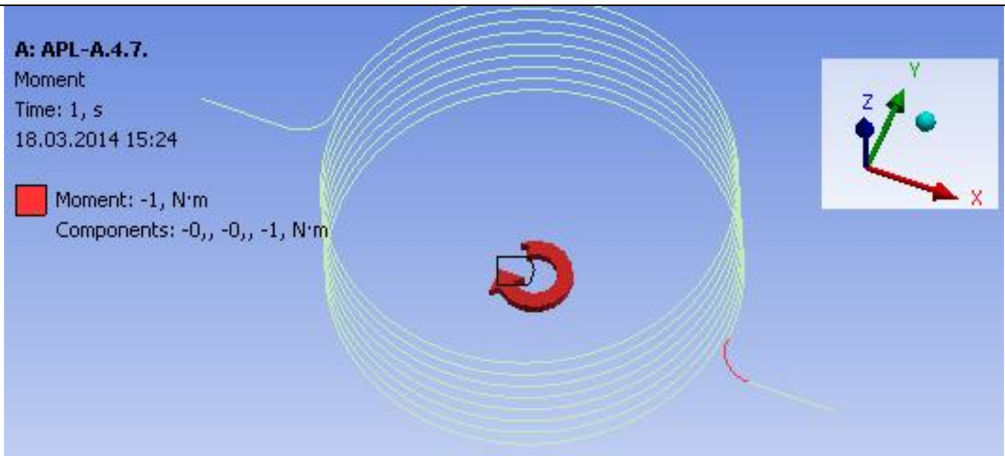


C.6 Load modeling

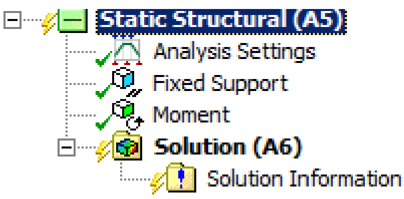
Input Remote Moment

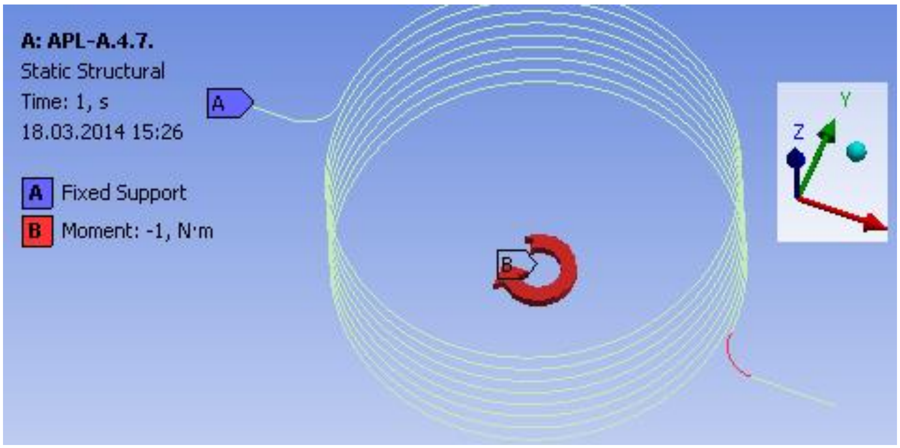
Outline: ↵ Static Structural (A5) → Loads → Moment → → Details of „Moment" → **Scope** → Geometry: [will be selected with ↵ the arc of the connecting circle between the spring propeller and the terminal segment at dimension 0, using the option (Edge)] → Apply; **Definition** → Magnitude: -1 Nm;

Moment → Promote Remote Point → ↵ Moment - Remote Point → Details of „Moment - Remote Point" → **Scope** → X Coordinate = 0, Y Coordinate = 0, Z Coordinate = 0 [the coordinates of the point P1 (0, 0, 0) made previously will be written].



The constraints and loads of the resort will look like the figure below





D. SOLVING THE FEA MODEL

D.1 Setting results

In order to select the final data types to be analyzed after the launch of the calculation module, follow the series of commands presented below.

→ ↳ Solution (A6) → Insert → Deformation → Total [use the commands in the open command box with ↳].

The same result can be obtained by using the commands:

, Outline: ↳ Solution (A6) → ↵ Insert → ↵Deformation → ↵ Total. [the buttons in the menu bars are used] and

↳ Solution (A6) → Deformation → Directional.

For this type of structure, the Beam tool can be applied in order to visualize the linearized stresses on the component elements. It is customary, in the process of designing bar structures, to take into account the components of axial stresses that arise from the effect of axial and bending loads in all directions. The following are the other types of results to be analyzed:

↳ Solution (A6) → Tools → Beam Tool.

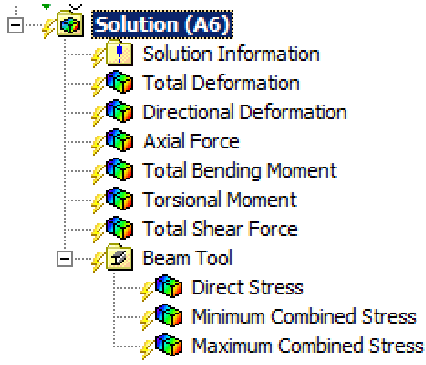
↳ Solution (A6) → Beam Results → Axial Force.

↳ Solution (A6) → Beam Results → Bending Moment.

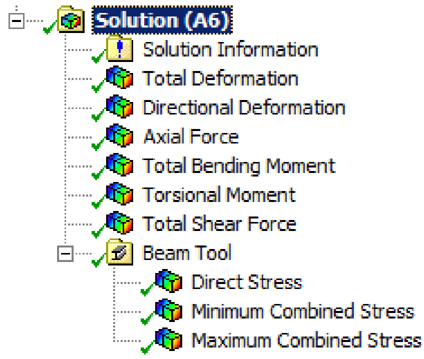
↳ Solution (A6) → Beam Results → Torsional Moment.

↳ Solution (A6) → Beam Results → Shear Force.

D.2 Lansarea modulului de rezolvare a modelului / Launching the solving module
Outline: ↵ Solution (A6) → Solve.



→ Solve →



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

For suggestive results, set the view scale of the menu bars:

Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

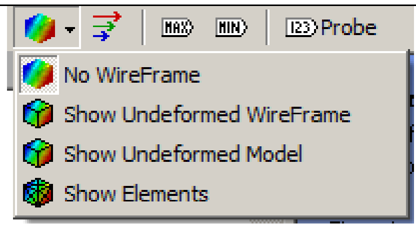
Total deformation viewing

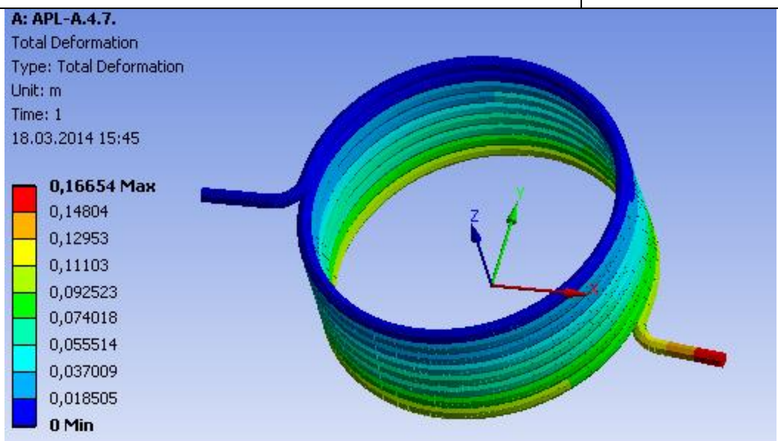
Outline: ↵ Solution (A6) → ↵ Total Deformation → Tab-ul Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation

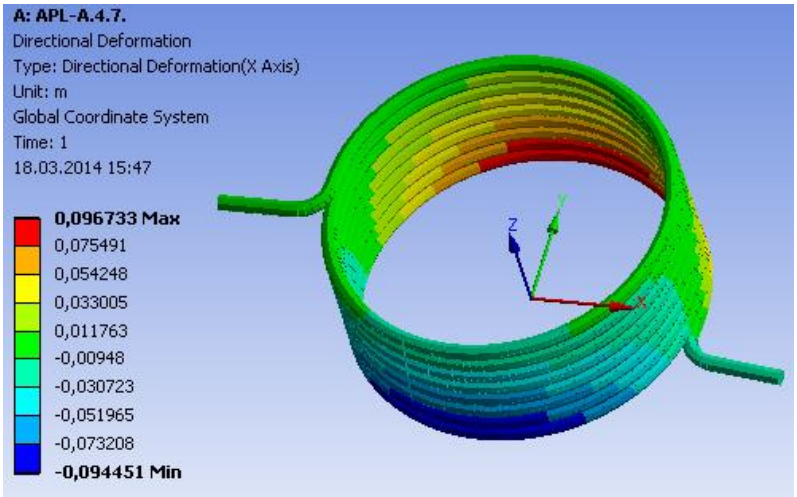
The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation 2 Sec (Auto). At the same time, the result can be saved as a video file using the *Export Video File* command.





Visualization of the deformation in one direction

Outline: ↵ Solution (A6) → Directional Deformation → Graph → Animation.



If you want to view it in another direction, follow the steps below:

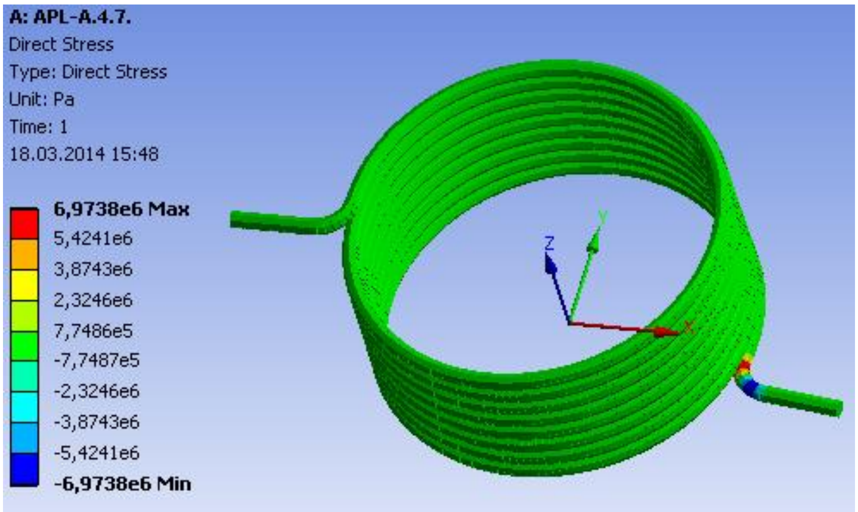
Outline: ↵ Solution (A6) → Directional Deformation → Details of „Directional Deformation” → **Definition** → Orientation : Y Axis → Solve.

E.2 Visualizing the fields of stresses, forces and moments

Direct stress

Direct Stress (σ_x) represents the component of the internal tension due to the axial force in the spring.

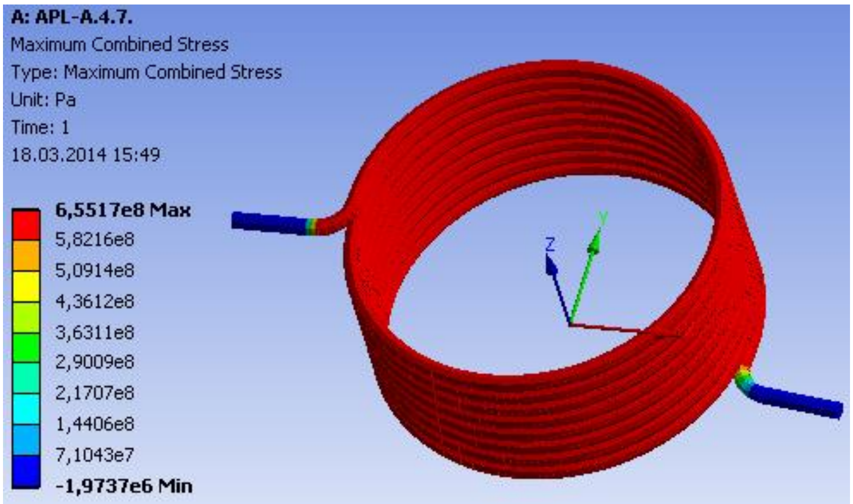
Outline: ↵ Solution (A6) → Direct Stress → Graph → Animation.

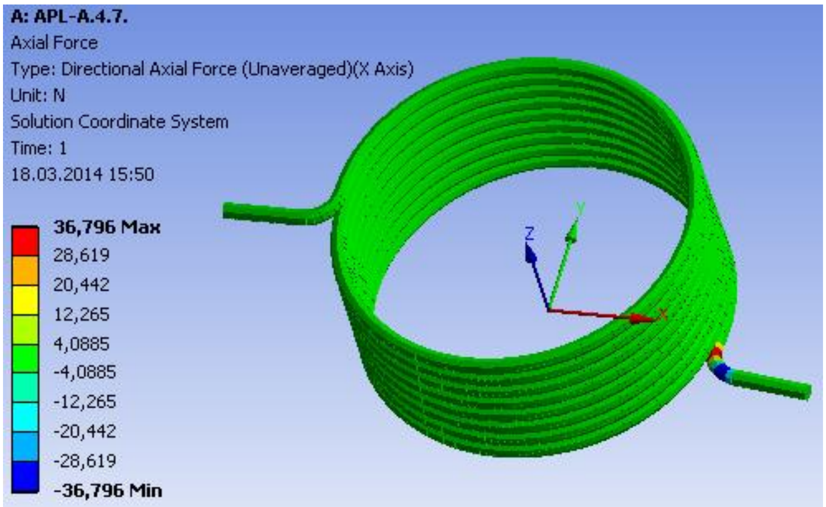
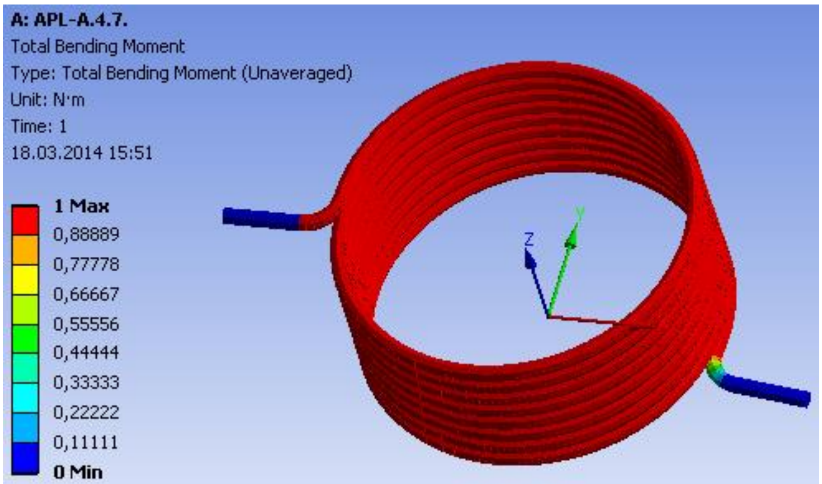
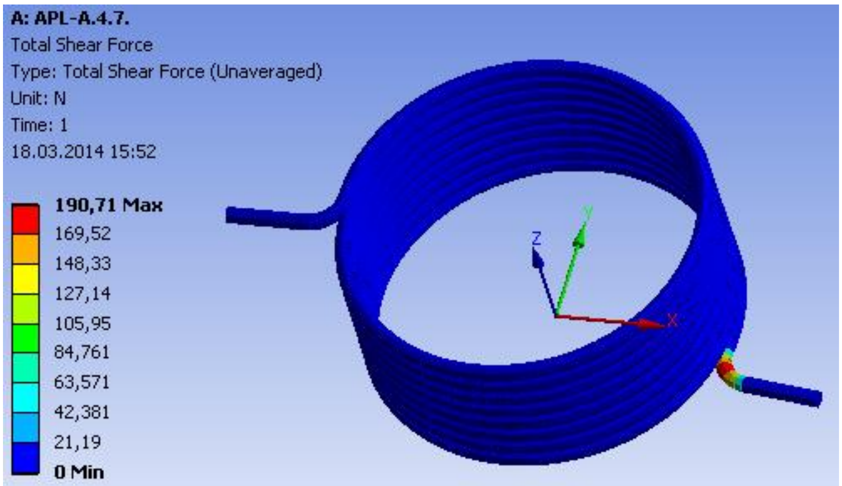


Maximum Combined Stress

Maximum Combined Stress –represents a linear combination of *Direct Stress* and *Maximum Bending Stress*.

Outline: ↵ Solution (A6) → Maximum Combined Stress → Graph → Animation.



Directional Axial Force , Outline: ↵ Solution (A6) → Axial Force → Graph → Animation .	
Bending Moment , Outline: ↵ Solution (A6) → Total Bending Moment → Graph → Animation .	
Shear Force , Outline: ↵ Solution (A6) → Total Shear Force → Graph → Animation .	

F. ANALYSIS OF RESULTS

F.1 Interpretation of results
It is observed that, despite the fact that the spring modeling was performed using a 1D body, the results obtained are suggestive, being presented in a 3D environment. From the point of view of the total deformations, it is observed that the maximum value is 166 mm, corresponding to the extremity of the segment in the drive area.

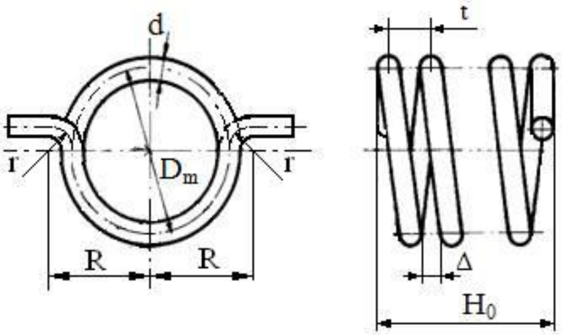
It is observed that the areas with high shear and bending efforts are those corresponding to the connection areas between the spring propeller and the right segment.
The information regarding the deformations, corroborated with the information regarding the internal stresses, the combined maximum stresses lead to the conclusion that the spring withstands loads without problems, the values of the maximum stresses not exceeding 6.5 x 10^8 Pa, value below the allowed material limit. Particular attention must be paid to the connections at the outlet of the spring propeller at both ends, these two areas being important concentrators of stresses.

F.2 Prezentarea rezultatelor obținute prin metoda clasică

Known geometric parameters:
• d = 2.5 mm - the diameter of the coil;
• Dm = 50 mm - average diameter;
• n = 8.5 turns;
• Δ = 0.5 mm, the clearance between turns.
Type of support area (support) and number of turns in this area: symmetrical outer hooks; connection radius, r = 2d; radius of action of the loading force, R = D_m / 2 + r.
Based on the constructive data of the spring in the figure above, the displacement and stiffness are calculated for a load M = 1,000 Nmm. The following values are obtained:

$$\theta_n = \frac{64\, M_{tn}\, D_m\, n}{E\, d^4} = 213{,}14 \text{ grd}$$

$$k = \frac{E\, d^4}{64\, D_m\, n}\, \frac{180}{\pi} = 4{,}7 \text{ Nmm/grd}$$



F.3 Comparative analysis of results

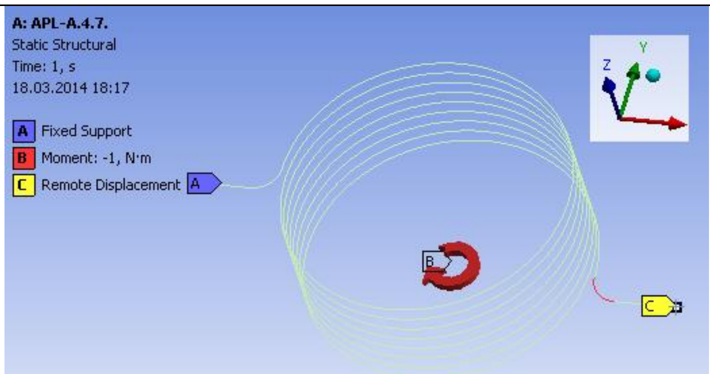
Using classical methods of Strength of Materials, the results are obtained by relatively simple calculations and can be compared with those obtained with MEF. On the other hand, by classical methods, very few results are obtained: only the angular displacement and the rigidity of the spring.

G. CONCLUZII / CONCLUSIONS

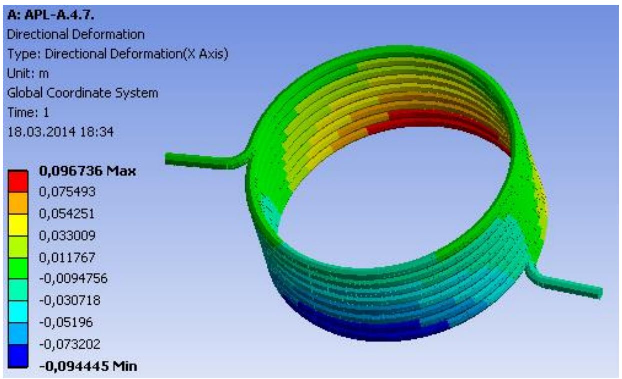
From the point of view of the pre-processing phase, it can be seen that the use of 1D bodies involves minimal resources for both modeling and discretization. Another strong point is that the profile of the spring can be modified / oriented very easily, without influencing the basic shape.
The introduction of supports, constraints and demands is quick and easy. The declaration of materials as well as discretization are controllable processes, which can be done automatically or manually.
Comparing the results obtained by the classical method and FEM, it can be seen that they are comparable, at least in the case of angular displacement, which was calculated classically, the finite element method providing much more data, over time and resource consumption much smaller.

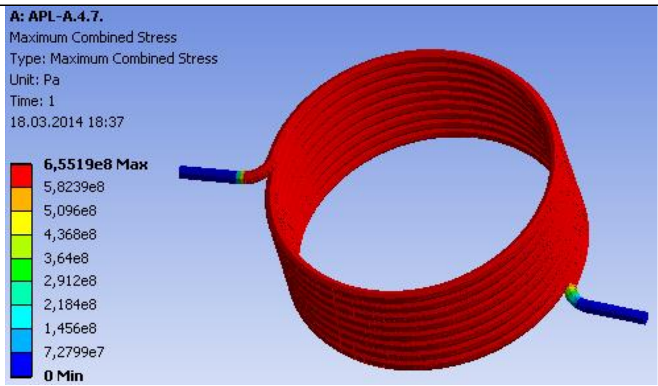
It can be seen that the spring is very strongly stressed in the connection area, at the exit of the propeller towards the extremities. The modification of these areas and the recalculation by FEM is done in a very short time, being an easy procedure. On the other hand, the model for analysis can be changed very easily, and it can change the supports and the demands very easily. In the case of geometries imported from other modeling programs, the geometric model will have to be modified in the original software, which will lead to the resumption of the procedure from the beginning.
For example, the analysis model can be modified by introducing an additional constraint, represented by the obligation for the final segment of the spring in the moment (force) request area to move in a plane. This means that the spring will not be deformed on the Oz axis.

, Outline: ↵ Static Structural (A5) → Supports → Remote Displacement → Details of „Remote Displacements” → **Scope** → Geometry: [will be selected with ↵ the end of the end segment of the spring located at dimension 0 mm, using the option (Vertex)] → Apply → Definition → X Component: Free, Y Component: Free, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.
The model for the analysis will look like this:



The results obtained for *Directional Deformation (X Axis)* as well as for *Maximum Combined Stress* are presented below.



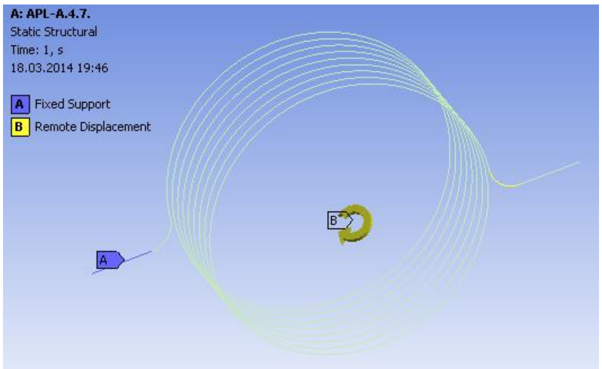


The results are almost identical to those obtained in the previous example. This is due to the fact that, by applying a moment via a Remote Point, the action of this request is required to take place only around the Oz axis, so it will only act in a plane parallel to xOy - equivalent to the newly imposed constraint in the second example.
Another model for analysis can be considered by replacing the moment applied to the spring with an imposed displacement of a certain angular value.
For this, the action of the moment will be suspended and an imposed angular displacement will be introduced.

M, Outline: ↳ Moment → Suppress;

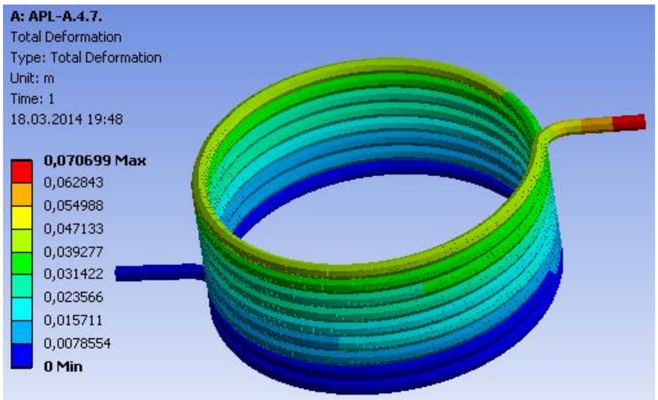
Supports → Remote Displacement →

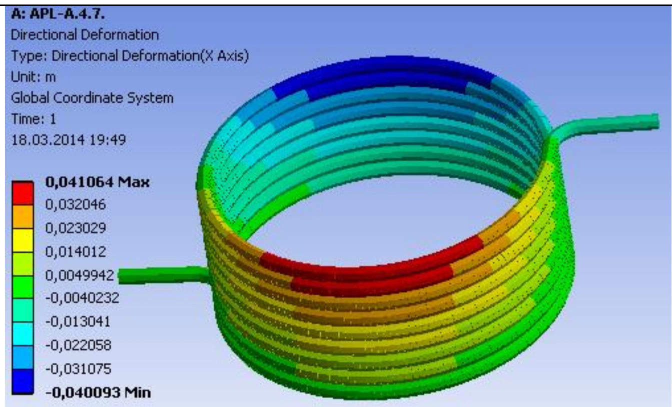
Details of „Remote Displacements" → **Scope** → Geometry: [will be selected with ↵ the arc of the connecting circle between the spring propeller and the terminal segment at dimension 0, using the selection filter (Edge)] → Apply → Definition → X Component: Free, Y Component: Free, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: -90°;

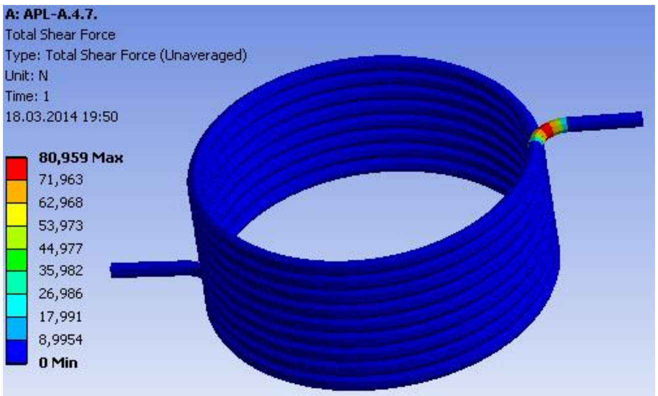


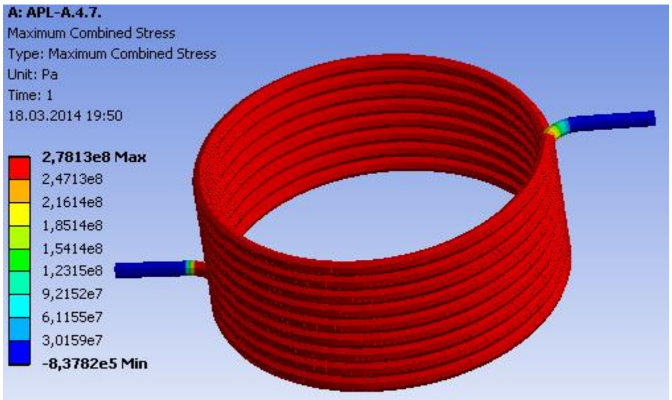
↳ Remote Displacement → Promote Remote Point → ↵ Remote Displacement - Remote Point → Details of „Remote Displacement - Remote Point" → **Scope** → X Coordinate = 0, Y Coordinate = 0, Z Coordinate = 0 [the coordinates of the point P1 (0, 0, 0) made previously will be written] → Solve.

The results obtained, for an imposed displacement of -90°, are presented below.









Application: FEA-A.13
Non-metallic elastic elements

KEY WORDS
Transient structural analysis, Nonlinear material, Hyperelastic material, 3D geometric model, 3D finite element, Damping elements, Own vibrations, Variable load

CONTENT
A. PROBLEM DESCRIPTION B. THE FEA MODEL C. PREPROCESSING OF THE FEA MODEL D. SOLVING THE FEA MODEL E. POSTPROCESING OF THE RESULTS F. ANALYZING OF THE RESULTS G. CONCLUSIONS

A. PROBLEM DESCRIPTION

A.1 Introduction

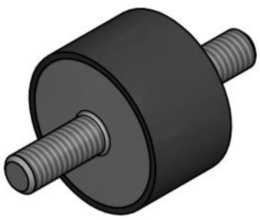
Many technical products contain mechanical elements that have distinct compact structures, required by the main function to be performed. Representative of this group of components are the elastic elements (springs), the damping elements, the housing support elements, etc. The specificity of these elements, as a rule, is given by their fixed or quasi-fixed connections with the neighboring parts.

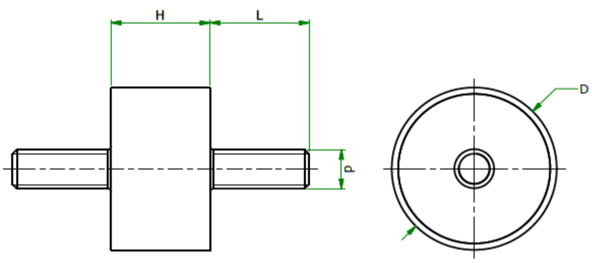
The finite element analysis of these components, in order to obtain precise results, presupposes the accurate definition of the solid model, of the restrictions imposed by the connections with the neighboring elements, as well as of the loads. This application will aim to perform a transient dynamic analysis to determine the response of the target structure to tasks that vary over time. If the effects of inertia and damping are not significant, another analysis can be performed - static structural with variable force.

A.2 Application description

Elastic elements made by rubber are frequently used in the construction of mechanical systems due to their elastic capacity and especially their damping capacity and, in many cases, due to their lower cost.

The buffer in the adjacent figure is composed of a cylindrical piece of rubber to which are attached two flat metal reinforcements.





These elements are frequently introduced in the subsystems of motor vehicles having the role of supporting parts or as elastic quasi-couplings with damping, bringing the following advantages: they eliminate wear and noise, dampen vibrations, have moderate costs and unpretentious maintenance. On the other hand, these elements have a shorter service life than steel due to the decrease in the strength and elasticity properties of rubber over time (aging process).

A.3 Application goal

The aim of this paper is to determine the displacement, deformation and stress fields of the standardized rubber buffer structure, type 497719, usually used to support the muffler in some vehicles. The rubber pad has the following dimensions: D = 70 mm, threaded rods M10, L = 43 mm, H = 70 mm. The loading and fixing of the studied element is done by means of the M10 metal rods integral with the flat metal reinforcements. The metal reinforcements are considered to be rigid and non-deformable in relation to the rubber mass of the element.

B. PREPARATION OF THE MODEL FOR ANALYSIS

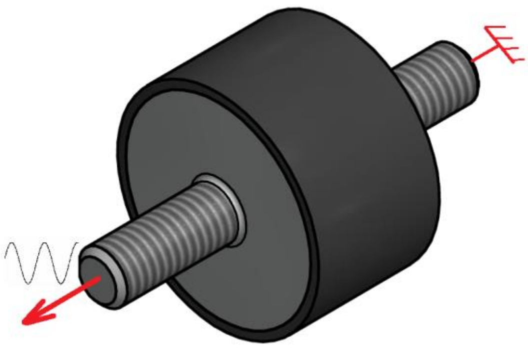
B.1 The model definition

In order to draw up the finite element analysis model associated with the above application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The geometric shape and dimensions of the rubber pad are shown in the adjacent figure. For the analysis, the structure of the buffer is modeled with 3D finite elements and, therefore, the geometric model is identical to the solid model. In order to simulate the behavior of the buffer as close to reality as possible, considering its loading by means of an M10 threaded rod and of the reinforcements made of steel, characterized by increased rigidities, the fixing constraints and loads will be introduced directly on the faces of the rubber cylinder. In order to simulate the connection with the outside by means of the external reinforcement, boundary conditions are introduced which imply translation restrictions after the three directions of the XYZ coordinate system for all points of the surface.



ITEM	D	H	d	L
497719	70	70	M10	43

B.3 Characteristics of the material
The analyzed element is made of neoprene rubber, with a hardness of 70 Sh and the following deformation constants: A_{10} = 0.177 N / mm2, A_{01} = 0.045 N / mm^2 and D1 = 333 N / mm^2. These characteristics, in the area of small deformations, correspond to the following physical parameters of analysis: • modulus of longitudinal elasticity, E = 400 MPa; • transverse contraction coefficient (Poisson), ν = 0.49.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Creating of the project

The following commands will be executed, in the order shown:

Toolbox: ↵ ⊟ Analysis Systems → ↵↵ Transient Structural (the subproject window appears automatically) → Save As... → File name: *Rubber element* → Save.

Problem type setting (3D)

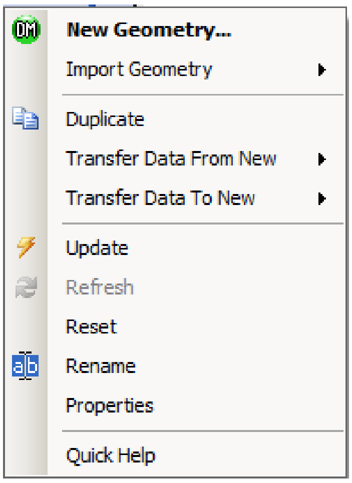
A : ↳ Geometry → ↵ Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵ Analysis Type, [select from list ↵▼, ↵ 3D] → [close window ↵ X].

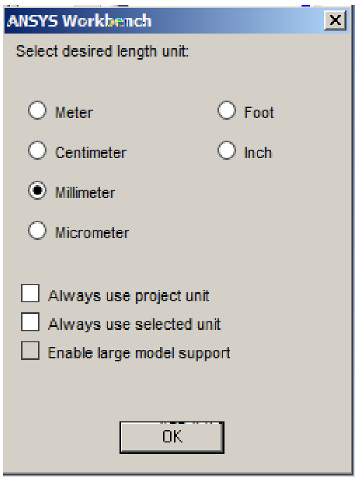
Save of the project

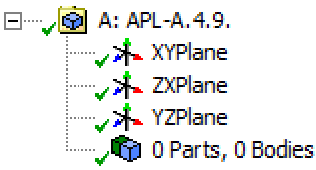
↵ Save As... → Save As, File name: [input name, *Rubber element*] → ↵ Save.

Setting the unit of measure for lengths

Toolbox: ↵ ⊟ Analysis Systems → **ANSYS Workbench**: Select desired length unit: Millimeter → ↵ (OK) → DM → Generate.





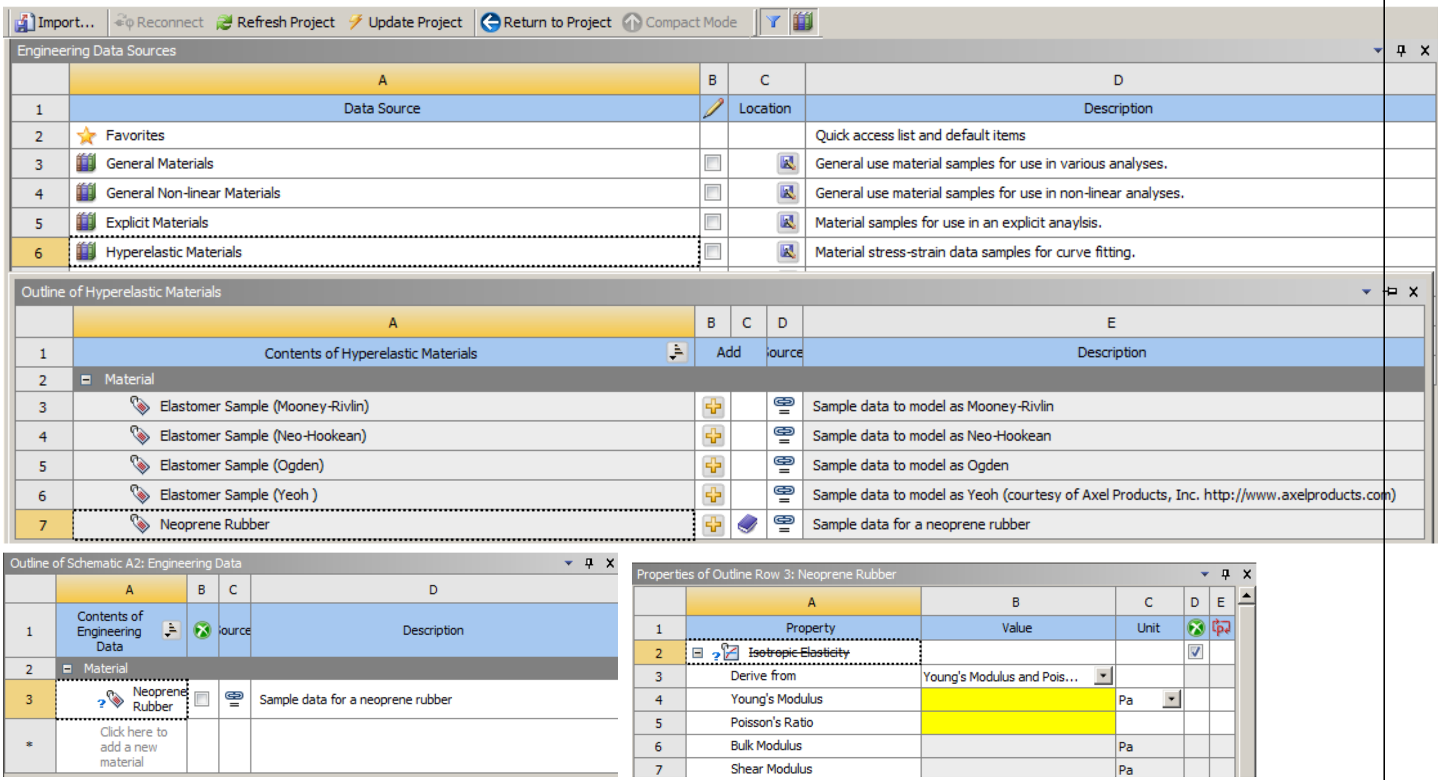


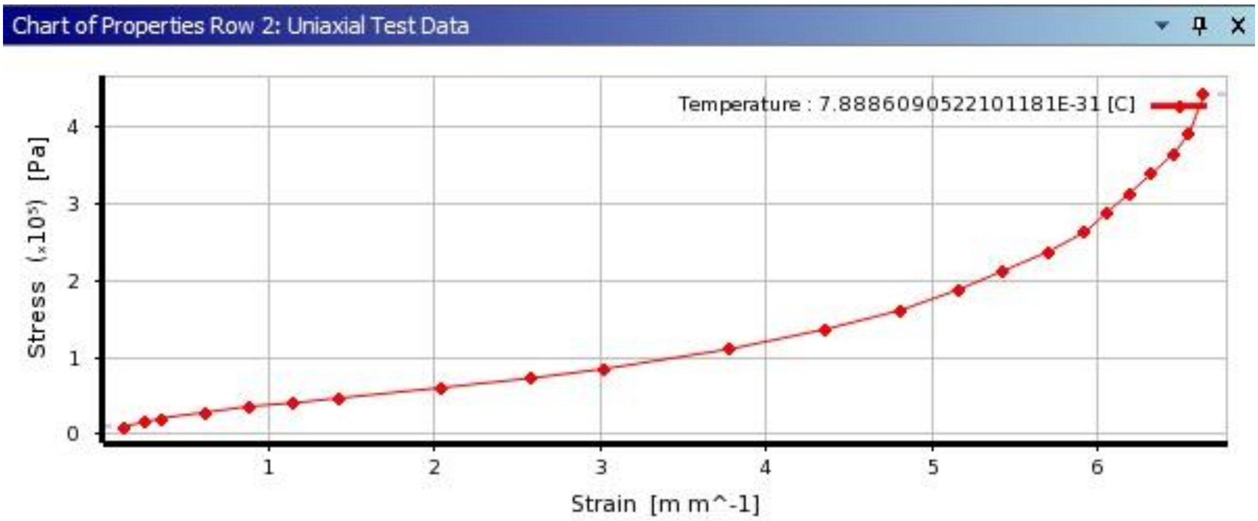
C.2 Modelling of material characteristics

Choosing of hyperelastic material (neoprene rubber):

→ ↳ Engineering Data → Edit... → Properties of Outline Row 3: Structural Steel (by default, the program opens the *Structural Steel* material, to be changed to *Neoprene Rubber*, chosen from the material database).

→ Outline of Schematic A2: Engineering Data → ↳ Structural Steel → Delete → ↵ → Engineering Data Sources → ↵ Hyperelastic Materials → Outline of Hyperelastic Materials → ↵ Neoprene Rubber → ↵ (Add to A2: Engineering Data) → Properties of Outline Row 7: Neoprene Rubber → the material characteristics stored in the program database are accepted → Update Project → Return to Project, except for a few properties: density (the value of 1000 kg / m^3 will be entered) as well as the modulus of longitudinal elasticity (E = 400 MPa) and the Poisson transverse contraction coefficient ($\nu = 0.49$).





C.3 *Creating the geometric model*

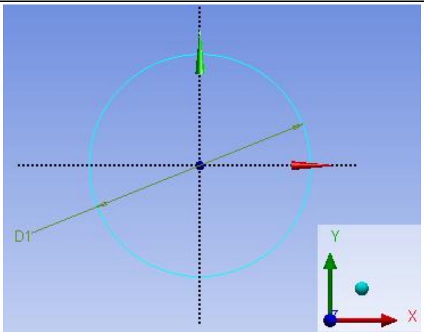
C.3.1 *Uploading DesignModeler Module (DM)*

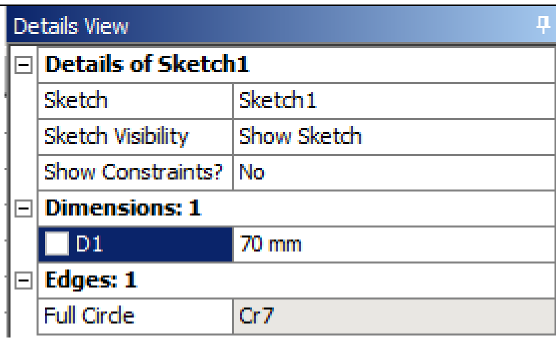
WB , Project Schematic: ↳ Geometry → ↵ DM New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.

C.3.2 *Creating the rubber component*

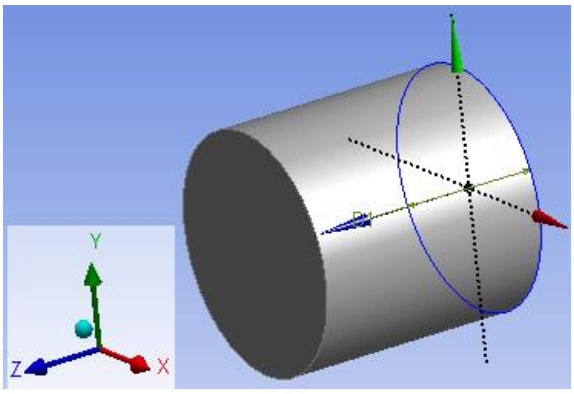
Creating the body

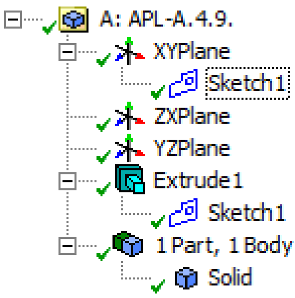
DM → Modeling → ↵ XYPlane → Sketching → → Draw → Circle [In the 2D modeling area a circle is created according to the sketch below] → Details View → **Dimensions: 1** → D1 = 70 → Generate.





DM → Modeling → Extrude → Details View → **Details of Extrude 1** → Geometry: Sketch1 → Depth: 70 → Generate





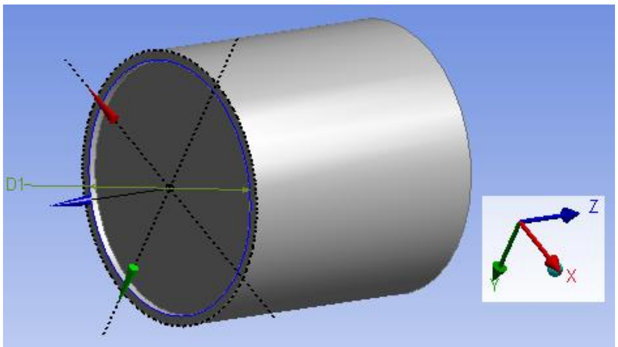
Making clearances at the ends

A 65 mm diameter and 2 mm deep recesses shall be drilled to simulate the position of the metal reinforcement of the buffer.

DM → Modeling → (New Plane) → Details View → **Details of Plane 1** → Type : From Plane → Base Plane: XY Plane → Generate → Sketching → → Plane 1 → Circle [In the plane P1 draw a concentric circle with circle D1 and diameter D2 = 65 mm] → Details View → **Dimensions: 1** → D1 = 65 → Generate.

DM → Modeling → Extrude → Details View → **Details of Extrude 2** → Geometry: Sketch2 → Operation : Cut Material → Depth: 2 → Generate

Repeat the above steps to clear the other end of the cylinder.



C.3.3 Saving the geometric model

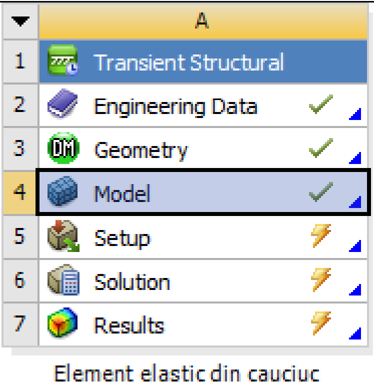
DM → (Save Project) → File → Close Design Modeler.

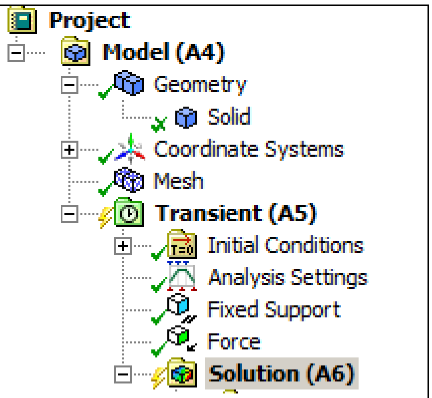
C.4 Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

→ ↵↵ Model → launching the module *Mechanical [ANSYS Multiphysics]*.

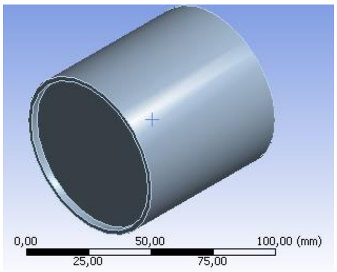
Outline: ↵ Solid → Details of "Solid" → **Material** → Assignment: Neoprene Rubber.





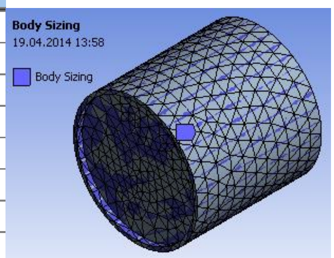
C.4.2 Model discretization and finite element size setting

Outline: ↵ Mesh → Mesh Control → Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ 3D body geometry, using the selection filter (Body)] Apply; **Definition Element** → Size: Default → Nonliniar Effects: Yes → Update. For a proper view of the discretization, select: ↵ Show Mesh.



Details of "Body Sizing" - Sizing

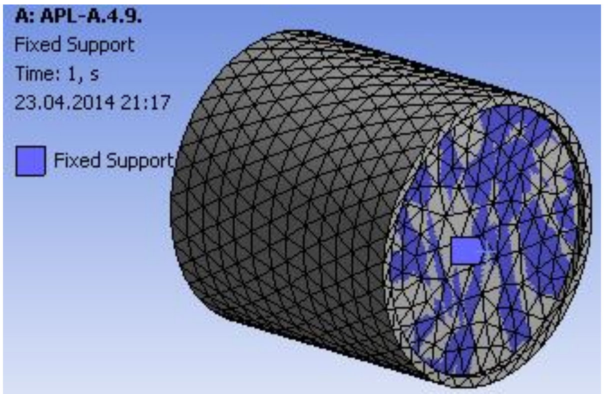
Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	Default
Behavior	Soft



C.5 Supports and restraints modelling

Input restraint

Outline: ↵ Static Structural (A5) → Supports → Fixed Support → Details of "Fixed Support" → **Scope** → Geometry: [the surface at the end of the spring at a height of 2 mm shall be selected with ↵ using the selection filter (Face)] → Apply.



C.6 Load modeling

Input forces

Because the part being used is used to secure a car's exhaust pipe, the stresses will not only be static, but will vary over time, depending on the vibration of the drum.

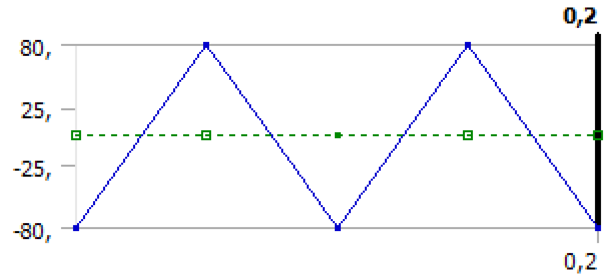
A sinusoidal, time-varying force-type load will be used in this study. Upload values will need to be entered in tabular form.

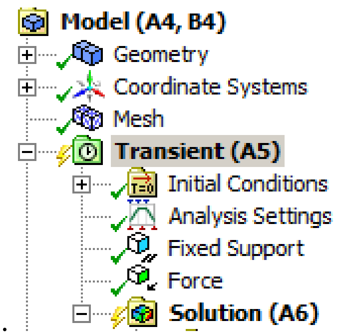
, Outline: ↵ ?Analysis Settings → Details of "Analysis Settings" → **Step Controls** → Number of Steps: 1, Current Step Number: 1, Step End Time: 0,5 s, Auto Time Stepping: Off.

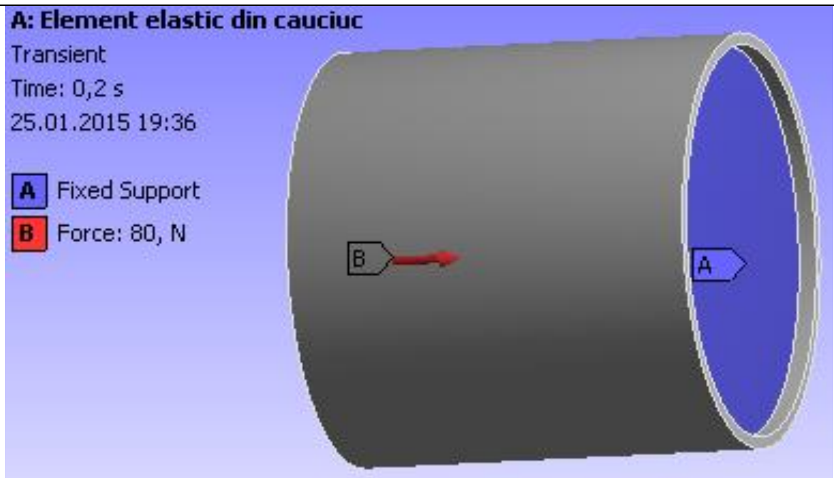
, Outline: ↵ Transient (A5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [selecting with ↵ the surface from the end of the buffer at distance 68 mm, using option (Face)] → Apply; **Definition** → Define by: Components; X Component = 0 N, Y Component = 0 N, Z Component: Tabular Data → Tabular Data → Tabular Data (the table with the values of the loads presented next will be completed). The constraints and loads of the resort will look like the figure below

Tabular Data

	Steps	Time [s]	X [N]	Y [N]	Z [N]
1	1	0,	= 0,	= 0,	-80,
2	1	5,e-002	= 0,	= 0,	80,
3	1	0,1	0,	0,	-80,
4	1	0,15	= 0,	= 0,	80,
5	1	0,2	= 0,	= 0,	-80
*					







D. SOLVING THE FEA MODEL

D.1 Setting results

In order to select the final data types to be analyzed after the launch of the calculation module, follow the series of commands presented below.

Total deformation setting

, Outline: ↳ Solution (A6) → ↵ Insert → ↵Deformation → ↵ Total.

Equivalent stress setting

↳ Solution (A6) → ↵ Insert → ↵ Stress → ↵ Equivalent (von-Mises).

One direction deformation settings

↳ Solution (A6) → Deformation → Directional.

Next, set the other types of results to be analyzed:

↵ Solution (A6) → Stress → Error.

↵ Solution (A6) → Strain → Equivalent (von-Mises).

↵ Solution (A6) → Energy → Strain Energy.

D.2 Launching the solving module

Outline: ↵ Analysis Settings → Details of "Analysis Settings" → **Solver Controls** → Large Deflection: On
↵ Solution (A6) → Solve.

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu bars:
Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

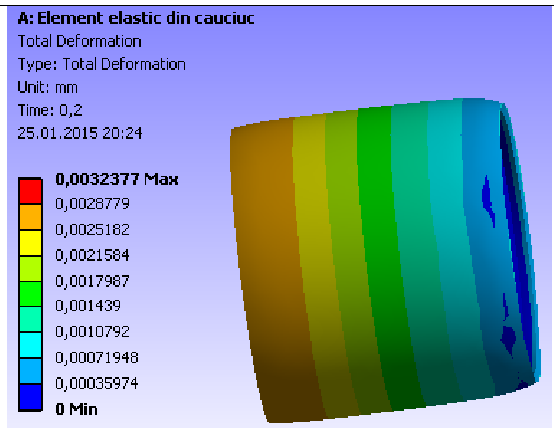
The section will be used to view the analyzed part in section (*New Section Plane*) located on the Desktop and a section plan will be chosen.

Total deformation view

↵ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation. The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command.

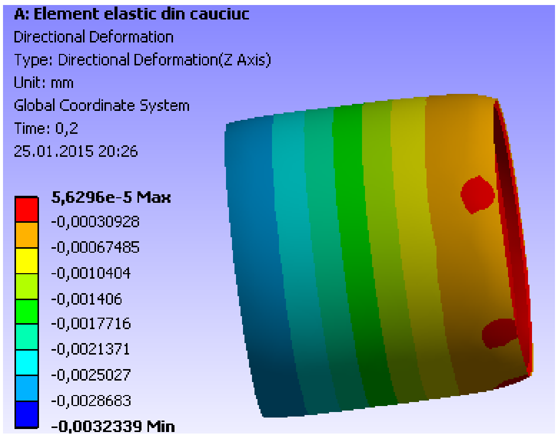


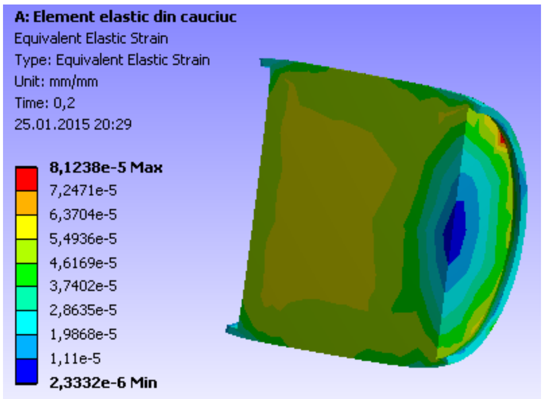
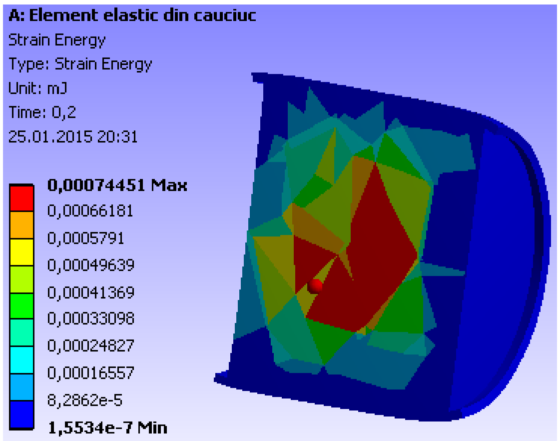
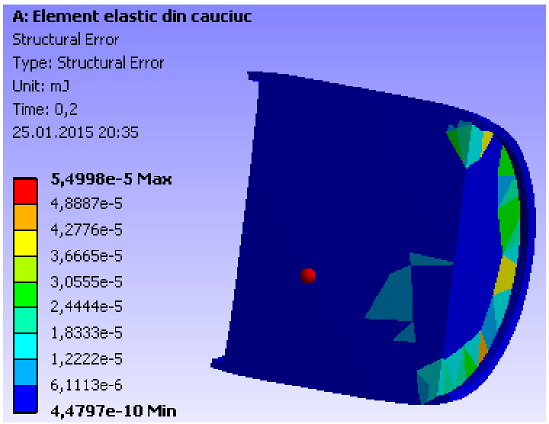
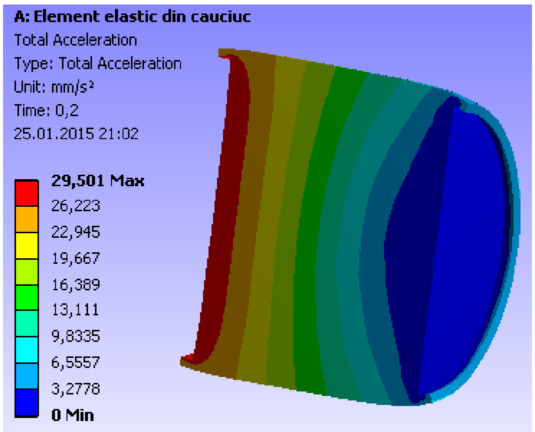
Visualization of the deformation in a certain direction

↵ Solution (A6) → Directional Deformation → Graph → Animation.

If you want to view in another direction, follow the steps below:

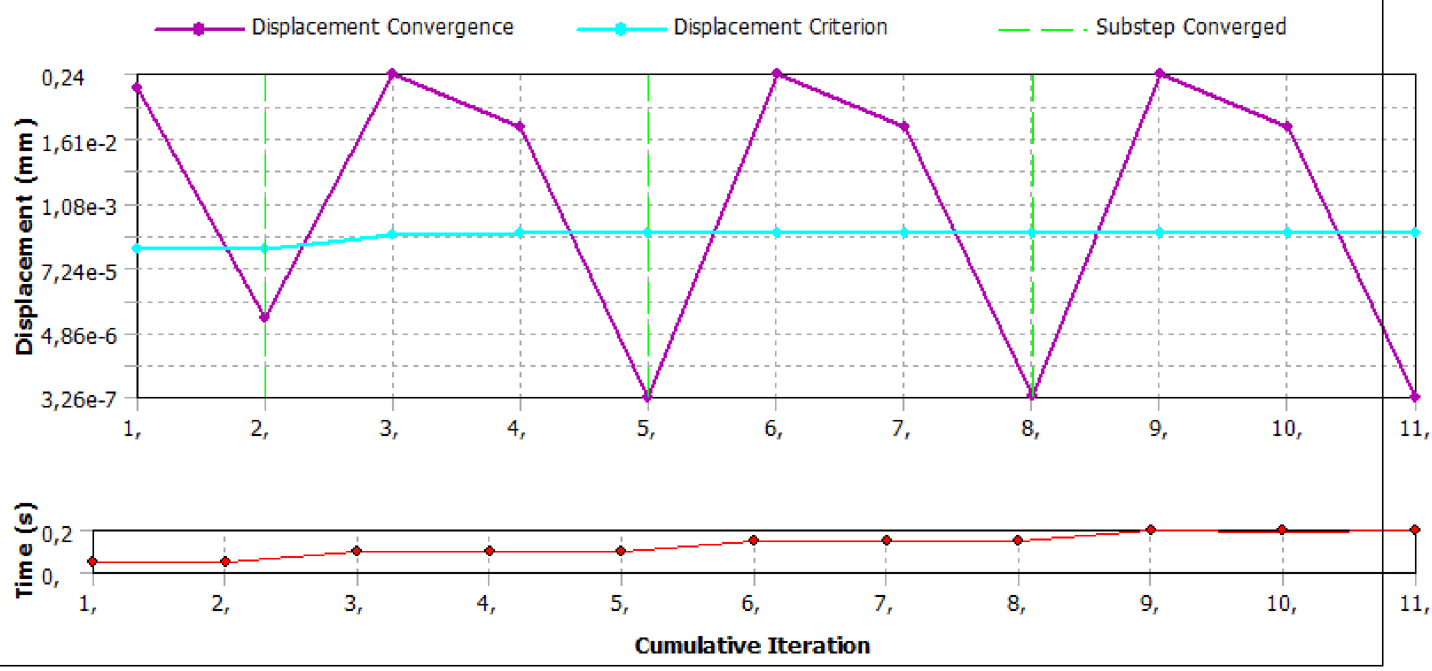
↵ Solution (A6) → Directional Deformation → Details of "Directional Deformation" → **Definition** → Orientation: Z Axis → Solve.



E.2 Visualization of equivalent stress fields, structural error and acceleration	
Equivalent Elastic Strain ↲ Solution (A6) → Equivalent Elastic Strain → Graph → Animation.	
Strain Energy ↲ Solution (A6) → Strain Energy → Graph → Animation.	
Structural Error ↲ Solution (A6) → Structural Error → Graph → Animation (sau Tabular Data).	
Total acceleration ↲ Solution (A6) → Total Acceleration → Graph → Animation (sau Tabular Data).	

E.3. Visualizing the convergence of solutions

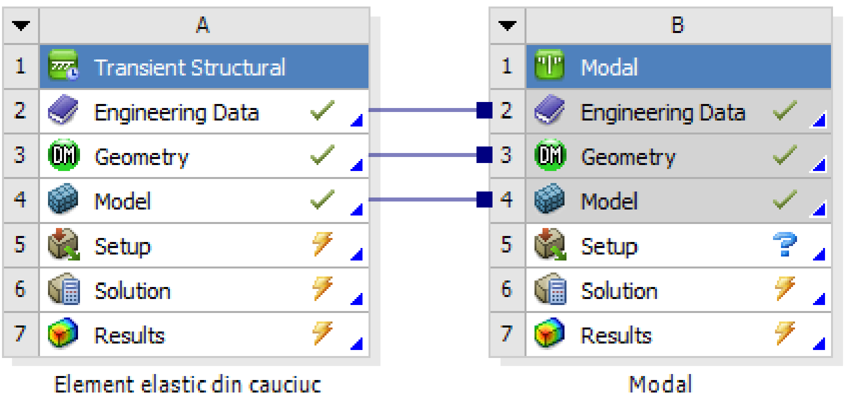
↲ Solution (A6) → Solution Information → Details of "Solution Information" → **Solution Informations** → Solution Output : Displacements Convergence.



F. ANALYSIS OF RESULTS

In order to avoid loading the buffer with a variable load that overlaps with its own vibration values, it is recommended to perform a modal analysis beforehand to determine them. For the rubber buffer, the modal analysis is performed as follows.

WB → Project Schematic → holding down ↲ on the command in the analysis structure ⊟ Analysis Systems pulls on the command Model from structure of *Transient Structural*, until it turns into Share B3, when the mouse key is released → OK



The natural vibration frequencies obtained by the modal analysis have very low values, far from the values of the stress frequencies induced by the operation of an internal combustion engine. From the point of view of deformations, it is observed that, following a variable stress in the range (-80, +80) N, they are in a very small value range, of the order of 0.005 mm, which does not cause a rapid deterioration. of the tampon.

	Mode	Frequency [Hz]
1	1,	0,
2	2,	0,
3	3,	0,
4	4,	1,8714e-004
5	5,	1,2895e-003
6	6,	1,6363e-003

G. CONCLUSIONS

Modeling and analysis with finite elements in this paper were made especially for teaching purposes following the user's initiation with the main stages of developing an FEA application in ANSYS Workbench, which emphasizes, in particular, the modeling and analysis of an elastic element made of -a material with nonlinear behavior (hyperelastic material - neoprene rubber).

The analysis algorithm for the Transient Structural type was highlighted, introducing time-varying stresses. At the same time, the importance of performing a modal analysis was highlighted in order to identify the values of the own vibrations, in order to be used in the design activity.

Application: FEA-A.14
Static analysis of beam structures

KEY WORDS
Linear static analysis, Linear material, 1D geometric model, 1D finite element, Linear finite element, Beam structures, Lattice beams, Comparison with classical methods

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

The beams are primary semi-finished products with one of the dimensions much larger than the other two have various constant sections (circular, annular, square, rectangular, profiles, etc.).

The structures made of beams are specific, especially, to metal constructions (bridges, beams, pillars, trusses, etc.). For finite element analysis, beam structures are modeled with one - dimensional finite elements whose properties are determined by dimensional and orientation sectional parameters. These models substantially reduce the memory requirement as well as the computation time. The results obtained from these finite element analyzes are less valid in the nodal connecting areas (welds, riveted joints, screw assemblies) which can be analyzed separately using 3D and connecting finite elements.

A.2 Application description

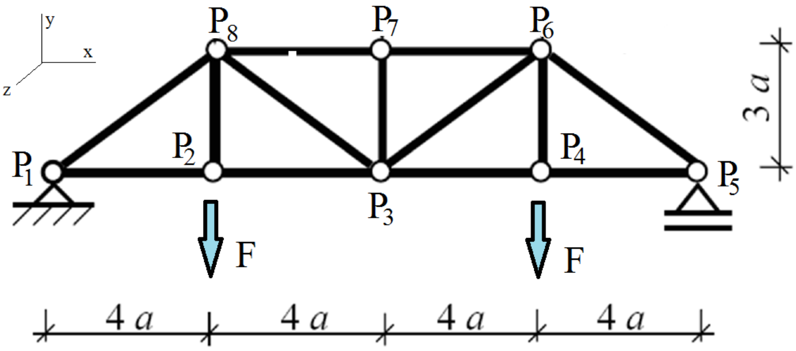
In order to support a water supply pipe when crossing a river, it is necessary to create a beam-type structure with lattice. The pipe is attached to the supports on the beam, placed at equal intervals, using clamps. In order to avoid the occurrence of thermomechanical stresses at temperature variations, the beam is fixed at one end by means of a bolt assembly that allows rotation and at the other end it is supported and guided allowing translation.

A.3 Application goal

In the case of this application, the analysis of the fields of displacements, deformations and stresses of a statically stressed beam structure is presented in order to optimize its construction, respectively to minimize its weight in compliance with the deformation and strength restrictions. For the beginning of the finite element analysis, the supporting structure in the figure above is considered to be made of square S235 steel pipe with dimensions 80 x 80 x 5 mm. The dimensions of the support structure are: length L = 16a = 8 m and height H = 3a = 1.5 m. It is considered that this structure supports, in the lower part, a pipe that is suspended by means of two flanges, at equal distances margins (l = 2 m). For finite element analysis, the action of the supported pipe on the structure can be modeled by

inserting in each node in which the pipe is attached by means of clamps a fixed force F = 5 KN. In addition, the consideration of internal forces of the own weight type is of particular importance for the analysis of these structures.


B. PREPARATION OF THE MODEL FOR ANALYSIS

B.1 The model definition
In order to draw up the finite element analysis model associated with the above application, it is necessary to identify: • geometric shape and dimensions, • restrictions induced by links with adjacent elements, • external and internal loads (own weight), • material characteristics.
B.2 The analysis model description
The geometric shape and the dimensions of the analysis model of the supporting structure are identical to those of the structure at the level of the sections of the sections. For the analysis, the structure is modeled with 1D finite elements and, therefore, the geometric model has the configuration in the figure below, having a = 0.5 m. In order for the analysis model to have the same behavior as the real model, it is necessary to associate limit conditions that imply the cancellation of the translational displacements in relation to the OX, OY and OZ axes and of the rotations in relation to the OX and OY axes, in point P1, respectively of the rotations OX, OY and of the translations along the axes OY and OZ, in point P5. The structure of the analysis model is loaded with concentrated force F = 5 kN at points P2 and P4.

B.3 Characteristics of the material
For finite element analysis the strength characteristics of the material, S235 steel (equivalent to OL 37) are: • modulus of longitudinal elasticity, E = 204,000 N / mm^2; • transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Creating of the project

WB, Toolbox: ↵ ⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [it can change the name Static Structural în *Structuri beame*].

Problem type setting (3D)

A: ↳ Geometry → ↵ Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵ Analysis Type, [select from list ↵▼, ↵ 3D] → [close window ↵ X].

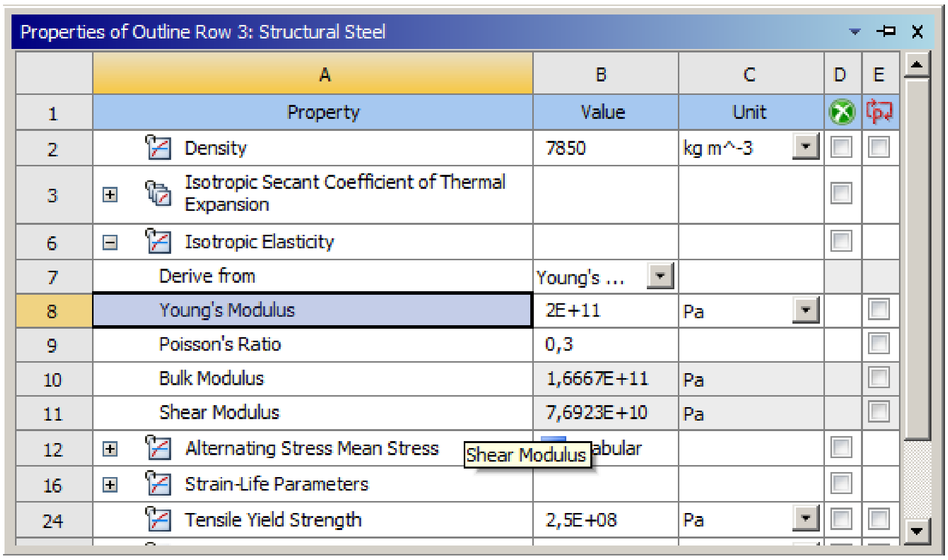
Saving of the project

↵ Save As… → WB Save As, File name: [input name, *Structuri beame*] → ↵ Save.

C.2 Modelling of material characteristics

WB, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit… → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, Young's Modulus, [select from list in column C (Unit) cu ↵▼, ↵MPa], [input in column B (Unit) valoarea / value, 204000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

If the window Properties of Outline Row 3: Structural Steel is not visible, the *Outline* and *Properties* or Reset Workspace options will be activated in the *View* menu.



Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	⊞ Isotropic Secant Coefficient of Thermal Expansion				
6	⊟ Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	⊞ Alternating Stress Mean Stress	Shear Modulus abular			
16	⊞ Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

C.3 *Creating the geometric model*

C.3.1 Uploading DesignModeler Module (DM)

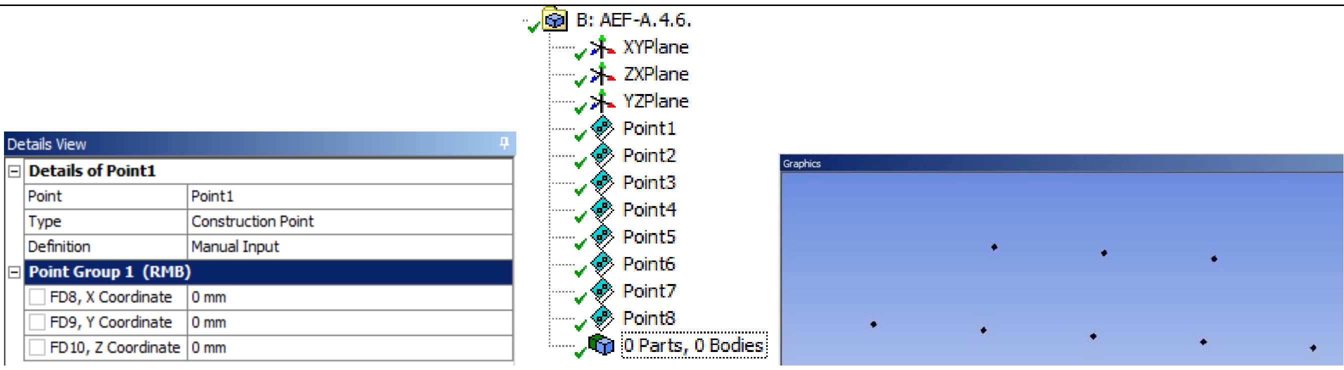
WB, Project Schematic: ↳ Geometry → ↵ DM New Geometry… → ANSYS Workbench: ↵ ◉ Millimeter, ↵OK.

C.3.2 Generating points

DM → Modeling → Create → Point [in the 3D modeling area the point P1 is created based on the Cartesian coordinates] → Details View → **Details of Point 1** → Definition ▼: Manual Input; **Point Group 1 (RMB)** → x = 0; y = 0; z = 0 → Generate.

The points P2… P8 are constructed in the same way, using the resulting Cartesian coordinates based on the dimensions given in the model for analysis:

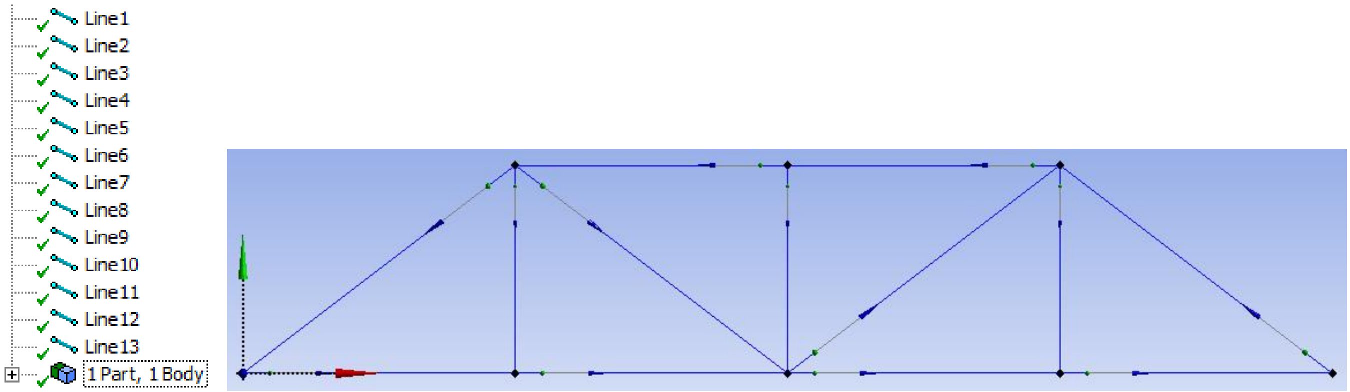
P2 (2000; 0); P3 (4000; 0); P4 (6000; 0); P5 (8000; 0); P6 (6000; 1500); P7 (4000; 1500); P8 (2000; 1500).



C.3.3 Generating of beam sections

DM → Modeling → <u>Concept</u> → Lines From Points → Details View → **Details of line 1** → Point Segments [while holding down the Ctrl key, select points P1 and P2]: Apply → Generate.

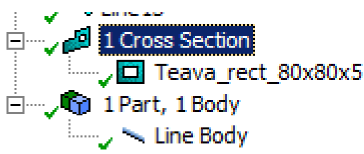
In the same way the segments corresponding to the pairs of points are constructed: (P2, P3), (P3, P4), (P4, P5), (P5, P6), (P6, P7), (P7, P8), (P8, P1), (P8, P2), (P8, P3), (P7, P3), (P6, P3), (P6, P4). The structure shown below will be obtained.

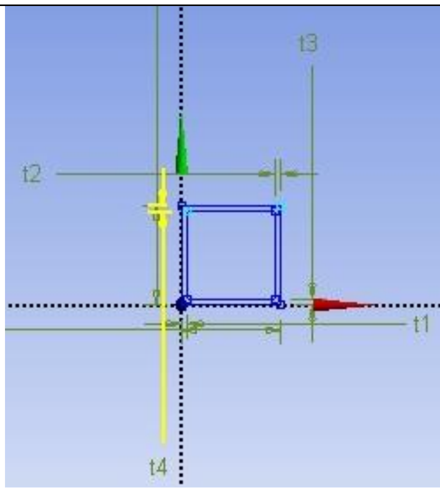


C.3.4 Segment section generation

DM → Modeling → <u>Concept</u> → Cross Section → Rectangular Tube → Details View → **Details of Rect Tube 1** → Sketch: Teava_rect_80x80x5;

Dimensions: W1 = 80 mm, W2 = 80 mm, t1 = 5 mm, t2 = 5 mm, t3 = 5 mm, t4 = 5mm → Generate.



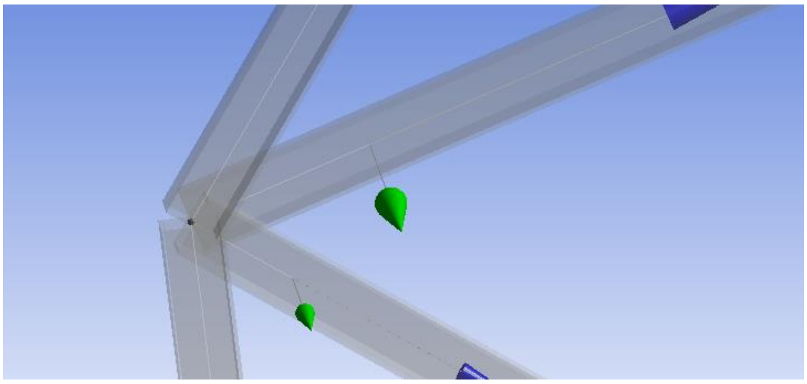


C.3.5 Assigning a transverse profile to the metal structure

Generating (building) a profile does not mean assigning it to a 1D beam structure. To complete the procedure, proceed as follows:

DM → ↵ Line Body → Details View → **Details of Line Body** → Cross Section: Teava_rect_80x80x5;

Modeling → <u>View</u> → Cross Section Solids [the option of 3D visualization of the sections is activated] → [check that the profiles are oriented correctly; the profile orientation is viewed using the green section view option is activated by green arrows; if the profile is not symmetrical and is not project oriented, the orientation can be changed accordingly: ↵ Line Body → Details View → **Line Body-Edge** → Reverse Orientation? : Yes] → Offset Type : Centroid → Generate.



C.3.6 Saving the geometric model

→ (Save Project) → File → Close Design Modeler.

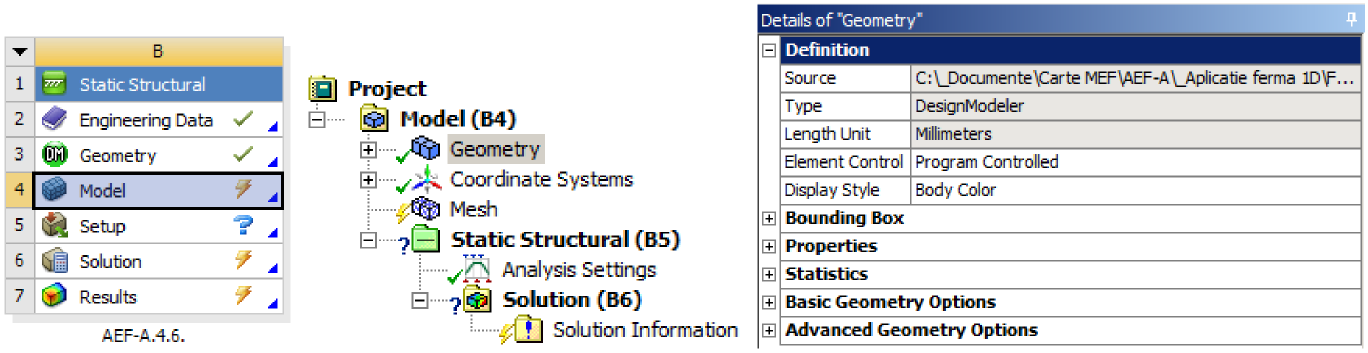
C.4 Finite element modelling

C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

Launching of the finite element modeling module

, Project Schematic ↵↵ Model → launching module *Mechanical [ANSYS Multiphysics]*.

↵ Geometry → Details of "Geometry" → **Definition** → Element Control : Program Controlled .



Setting the unit of measurement system

: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

Setting the material characteristics

Outline: ↵ Geometry → ↵ Details of "Geometry" → **Material** : ↵Assignment → [select from list ↵ , ↵ Structural Steel (default)].

Setting the model type

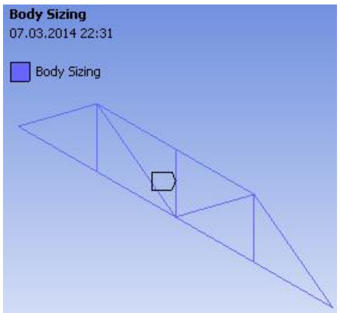
Outline: ↵ Geometry → Details of "Geometry", **Definition** : ↵3D Behavior.

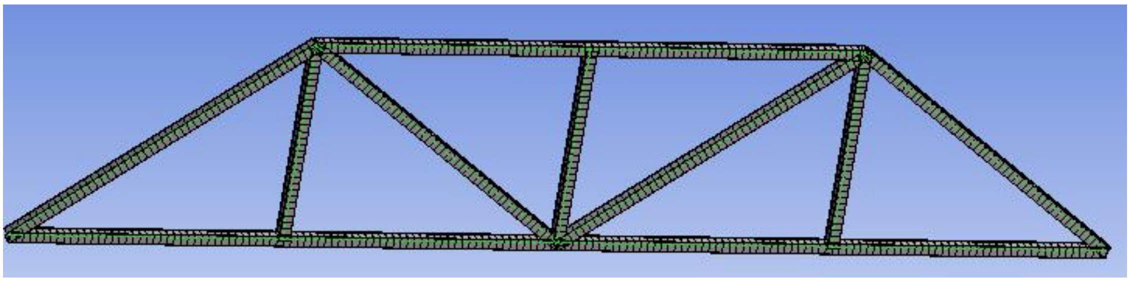
C.4.2 Model discretization and finite element size setting

, Outline: ↵ Mesh → Mesh Control → Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ structure geometry, using the selection filter (Body)] Apply; **Definition Element** → Size: 0,05 m → Update

Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	5,e-002 m
Behavior	Soft

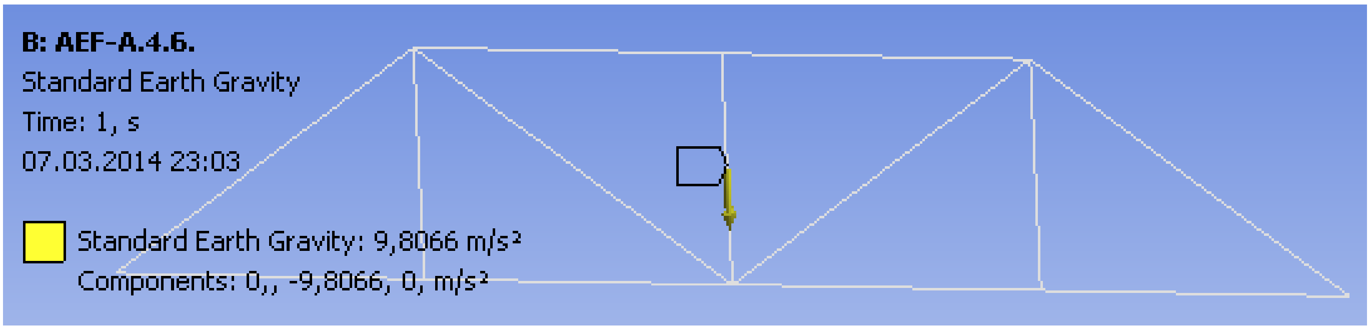




C.5 Supports and restraints modelling

Introduction of gravitational acceleration

→ Static Structural (B5) → Inertial → Standard Earth Gravity (the selection of the gravitational acceleration implies the taking into account of the own weight of the metallic structure) → Details of "Standard Earth Gravity" → **Definition** → Direction : -Y Direction.



Input restraint

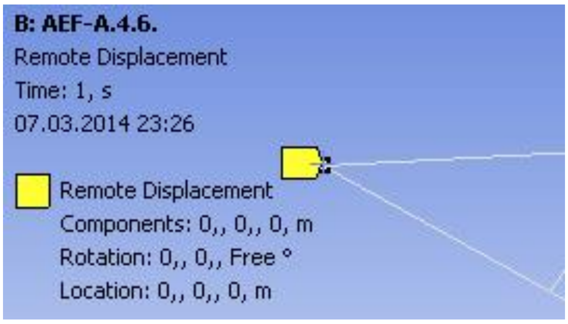
→ Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" →

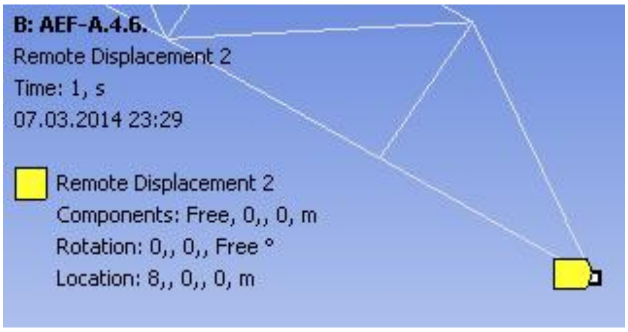
Scope → Geometry: [will be selected with ↵ point P1, using the option (Vertex)] → Apply; **Definition** → X Component: 0, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.

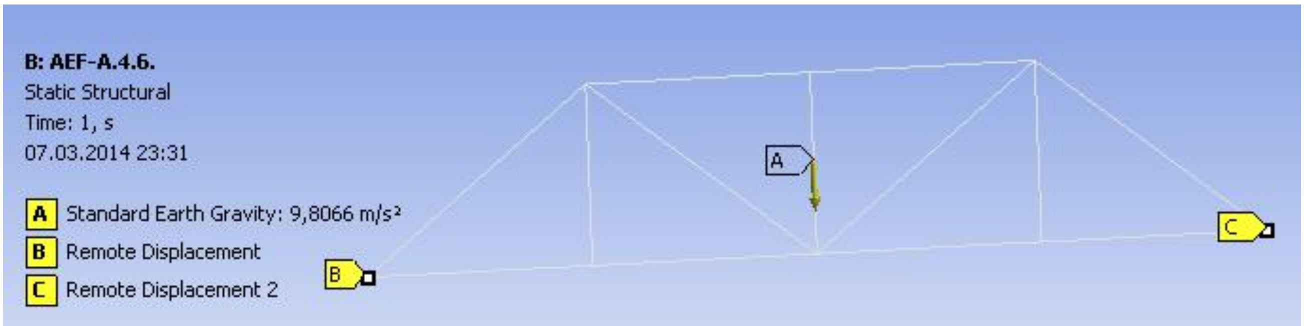
The procedure will be repeated for point P5:

→ Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" →

Scope → Geometry: [will be selected with ↵ point P5, using the option (Vertex)] → Apply; **Definition** → X Component: Free, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.







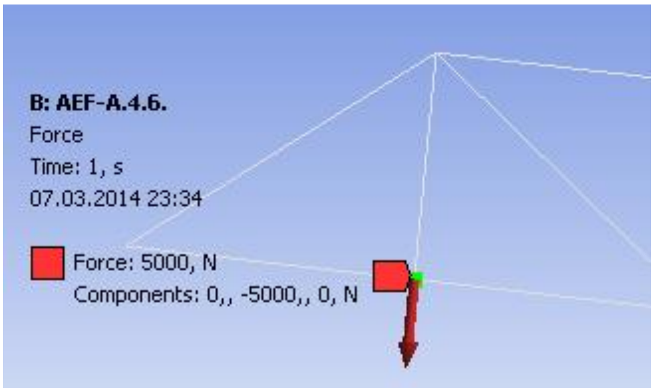
C.6 Load modeling

Input forces in selected point

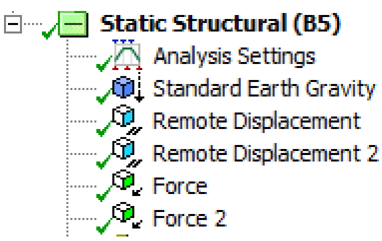
→ ↵ Static Structural (B5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [select point P2 using option (Vertex)] → Apply; **Definition** → Magnitude: 5000 N; Direction: axa Y [a segment of the metal structure parallel to the OY axis will be selected with ↵].

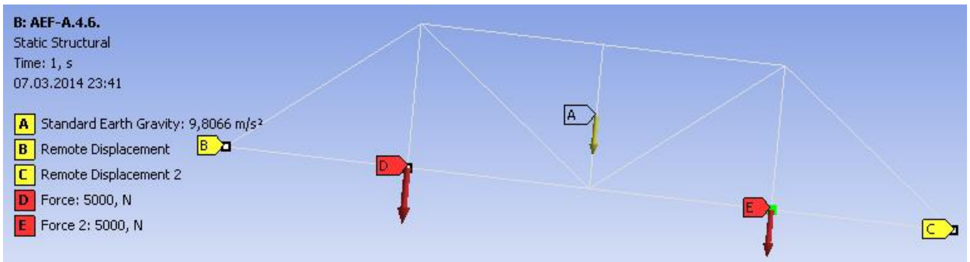
The procedure will be repeated for point P4:

→ ↵ Static Structural (B5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [select point P2 using option (Vertex)] → Apply; **Definition** → Magnitude: 5000 N; Direction: axa Y [a segment of the metal structure parallel to the OY axis will be selected with ↵].



The constraints and loads of the structure will look like the figure below.





D. SOLVING THE FEA MODEL

D.1 Launching the calculation module and select the types of results

In order to select the final data types to be analyzed after the launch of the calculation module, the series of commands presented below will be followed. → ↳ Solution (B6) → Insert → Deformation → Total [use the commands in the open command box with ↳].

Acelaşi rezultat se poate obţine prin utilizarea comenzilor:

↲ Solution (B6) → Deformation → Total [the buttons in the menu beams are used]

and

↲ Solution (B6) → Deformation → Directional.

For this type of structure, the Beam tool can be applied in order to visualize the linearized stresses on the component elements. It is customary, in the process of designing beam structures, to take into account the components of axial stresses that arise from the effect of axial and bending loads in all directions. The following are the other types of results to be analyzed:

↲ Solution (B6) → Tools → Beam Tool.

↲ Solution (B6) → Beam Results → Axial Force.

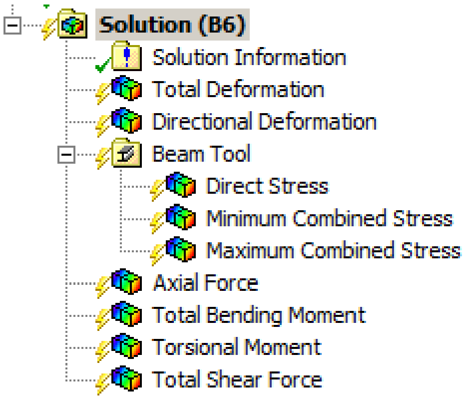
↲ Solution (B6) → Beam Results → Bending Moment.

↲ Solution (B6) → Beam Results → Torsional Moment.

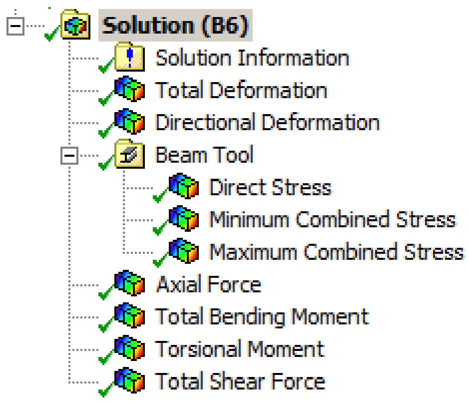
↲ Solution (B6) → Beam Results → Shear Force.

D.2. Launching the solving module

→ ↲ Solution (B6) → Solve.



→ Solve →



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu beams:

Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

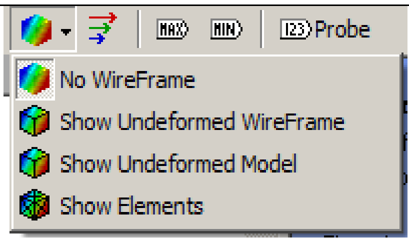
Total deformation view

↲ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

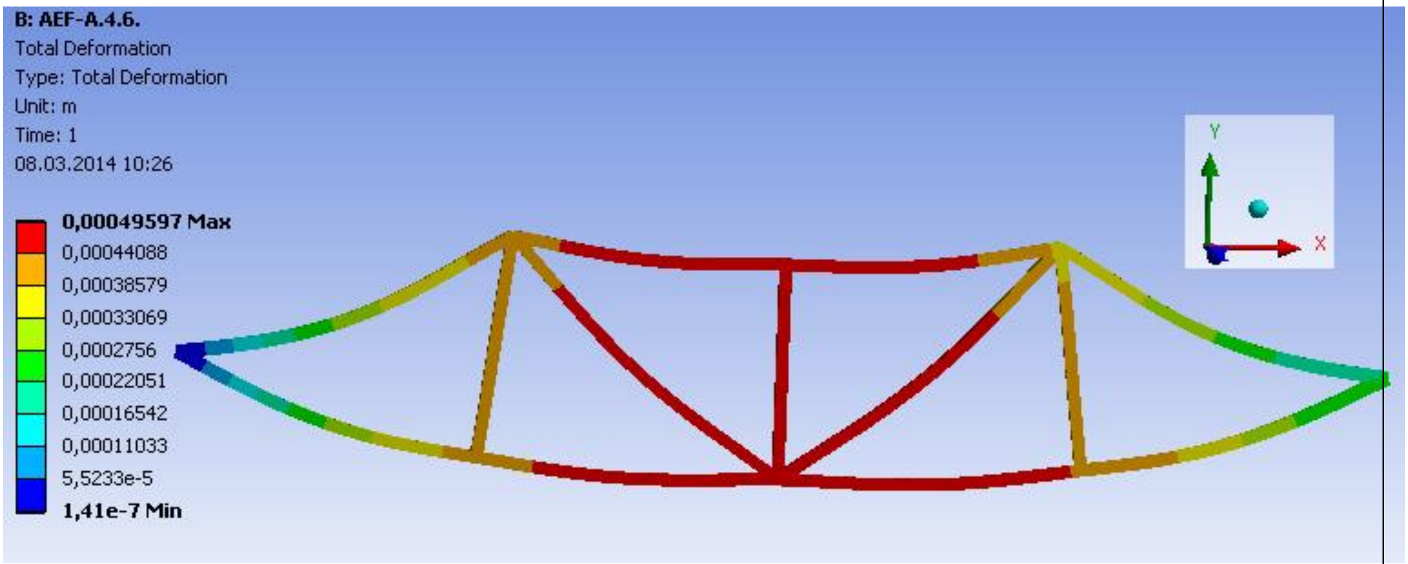
Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation.

The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command.



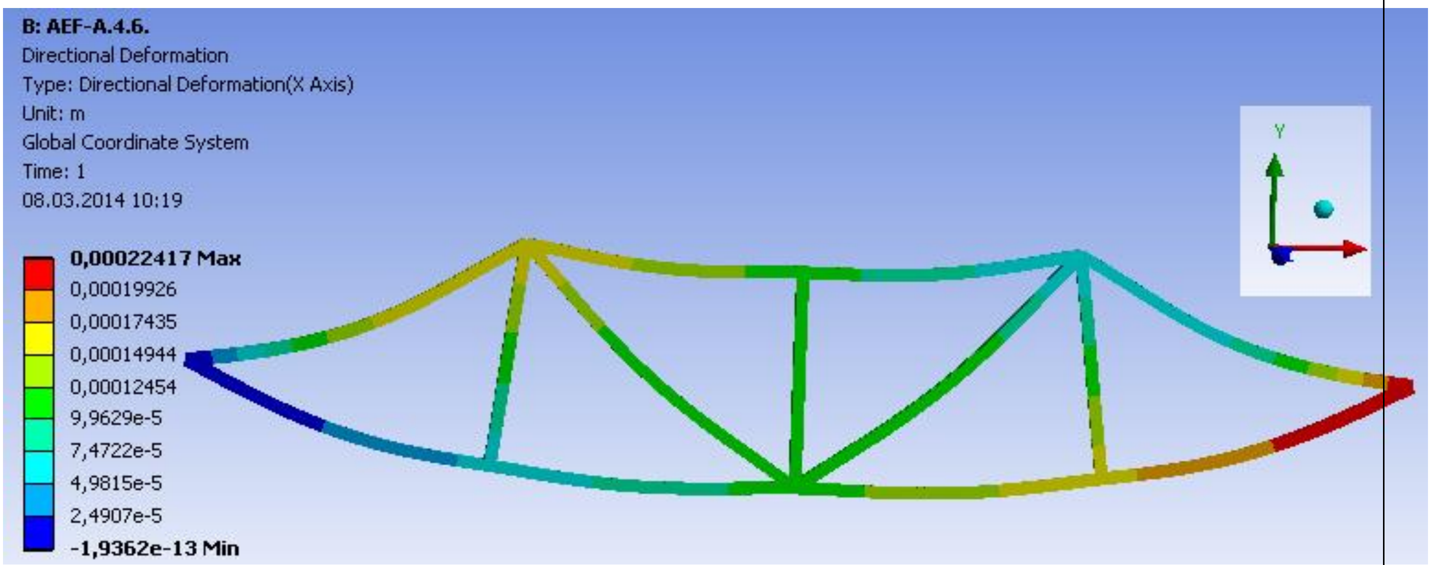
Vizualizare deformației totale / Total deformation view

↵ Solution (B6) → Total Deformation → Graph → Animation.



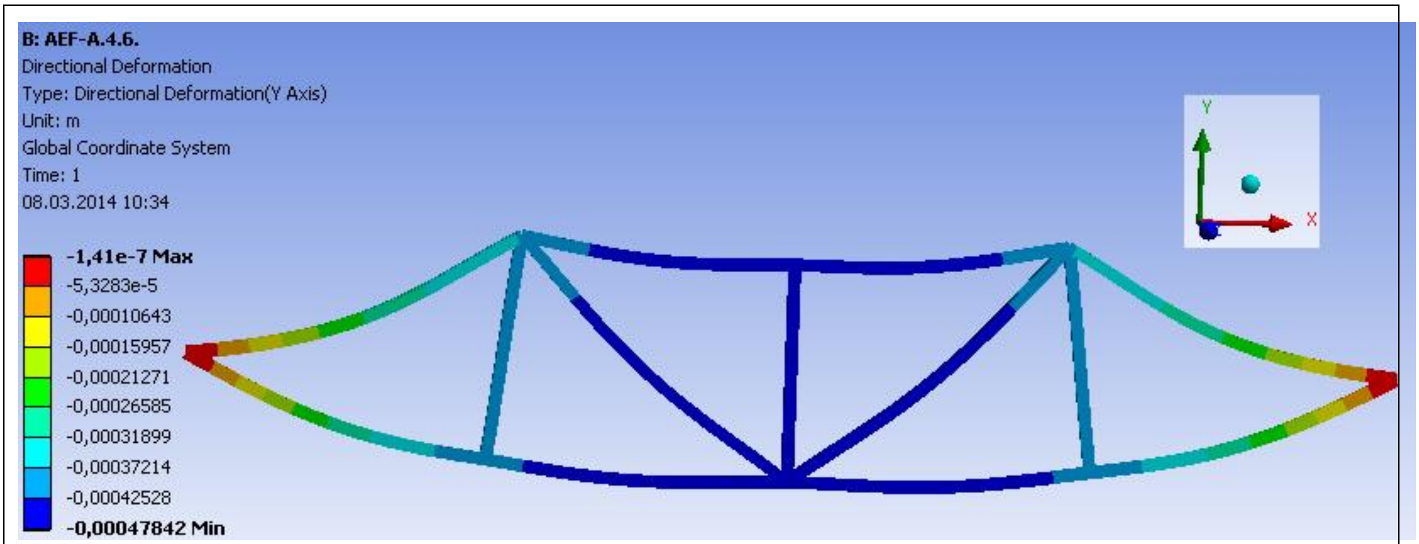
Visualization of the deformation in a certain direction

↵ Solution (B6) → Directional Deformation → Graph → Animation.



If you want to view in another direction, follow the steps below:

↵ Solution (B6) → Directional Deformation → Details of „Directional Deformation" → **Definition** → Orientation : Y Axis → Solve.

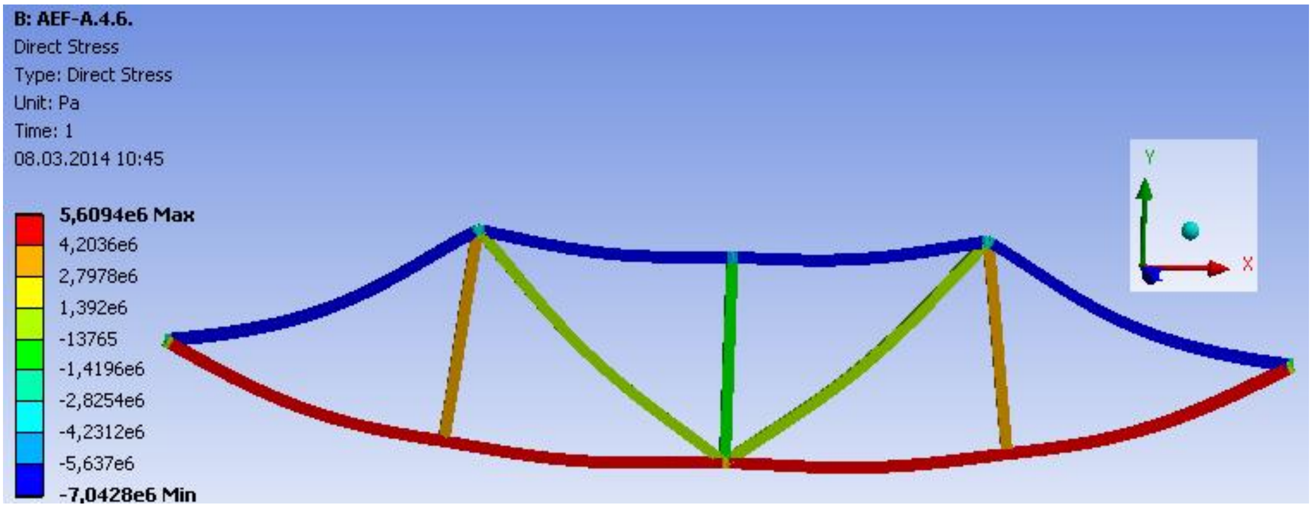


E.2. Visualizing the fields of stresses, forces and moments

Direct Stress

Direct Stress (σ_x) represents the component of the internal tension due to the axial force in a section of the beam.

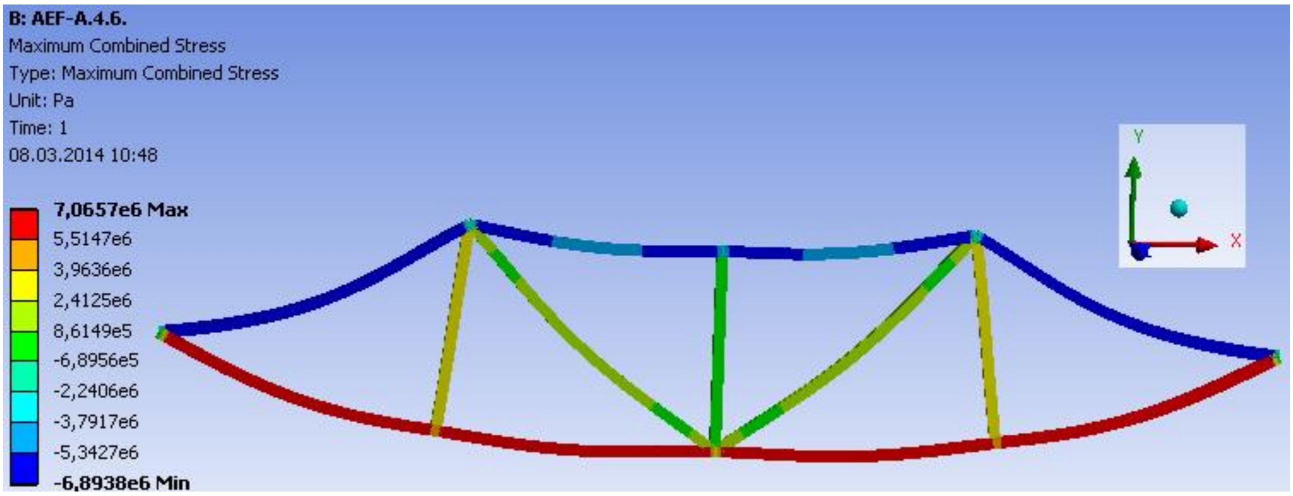
↲ Solution (B6) → Direct Stress → Graph → Animation.



Maximum Combined Stress

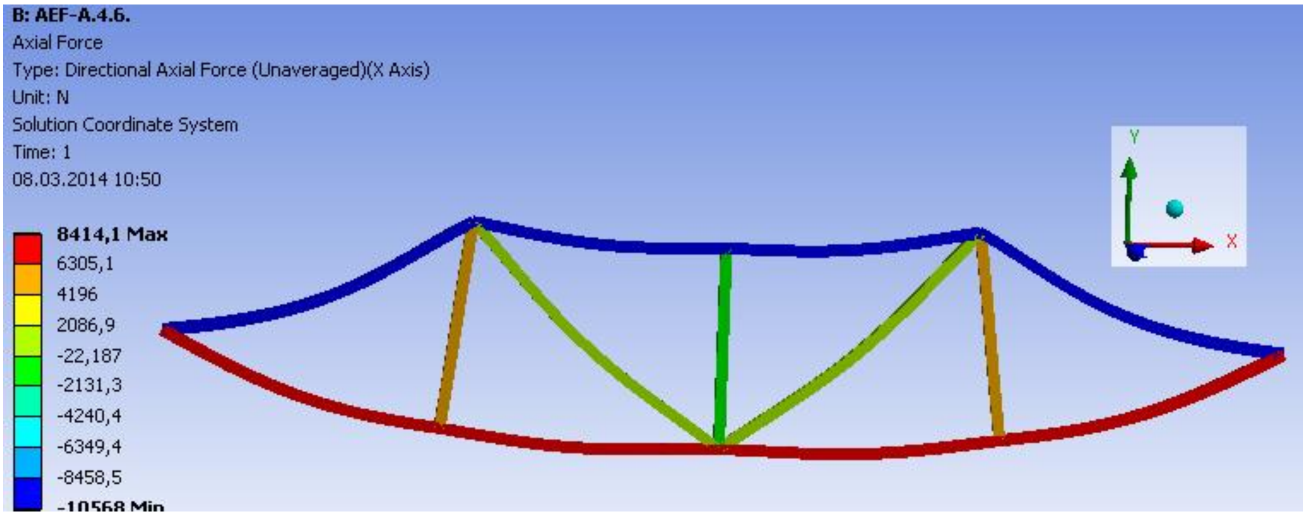
Maximum Combined Stress – represents a linear combination of *Direct Stress* şi / and *Maximum Bending Stress*.

↲ Solution (B6) → Maximum Combined Stress → Graph → Animation.



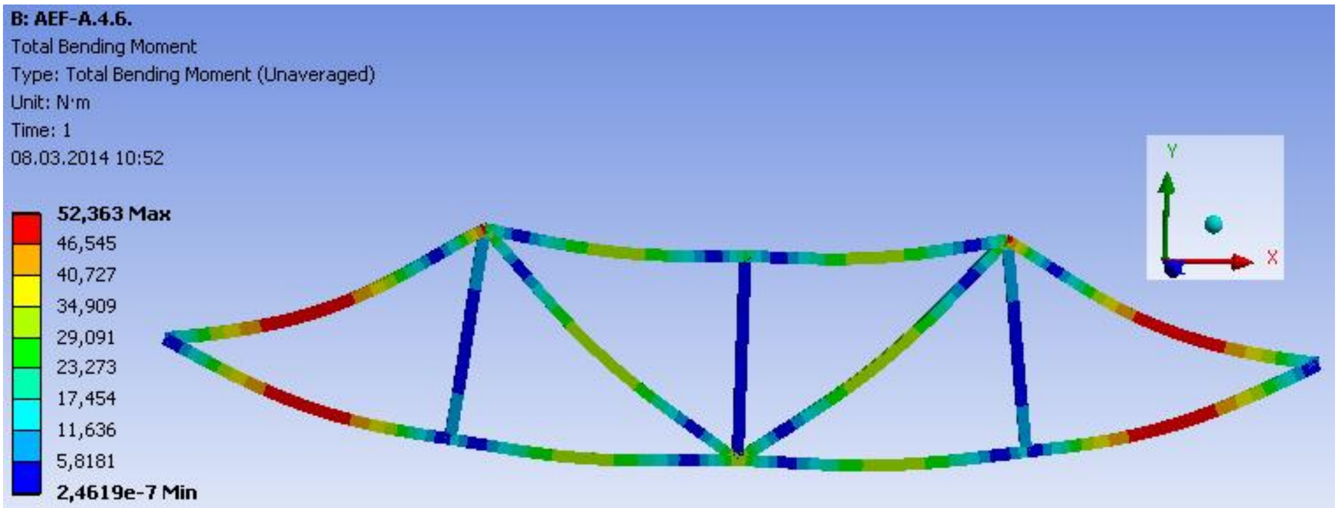
Directional Axial Force

Solution (B6) → Axial Force → Graph → Animation.



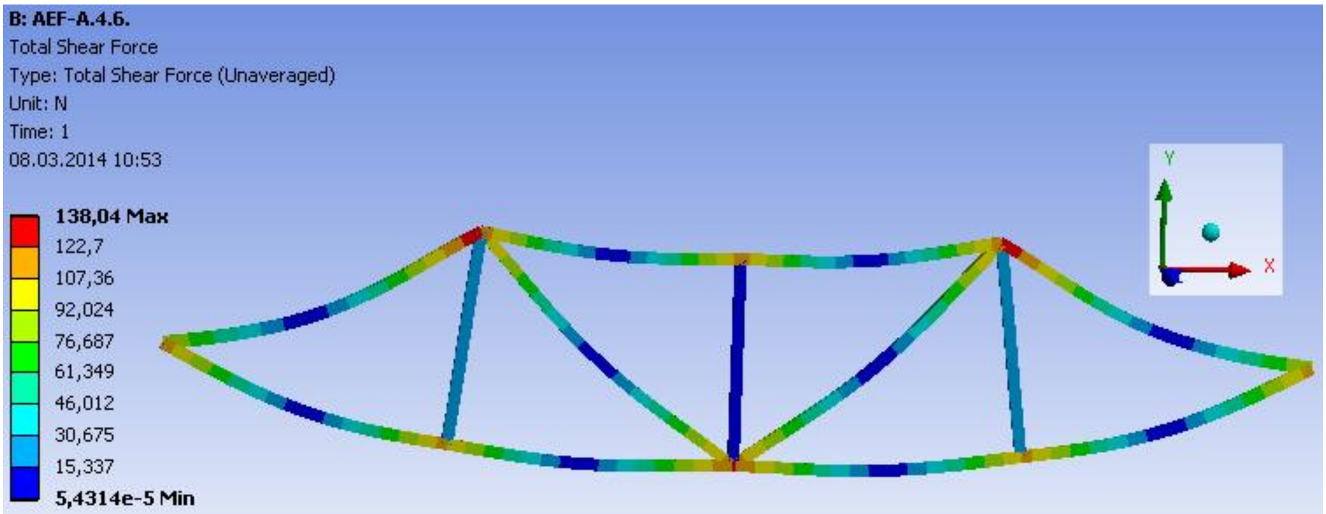
Bending Moment

Solution (B6) → Total Bending Moment → Graph → Animation.

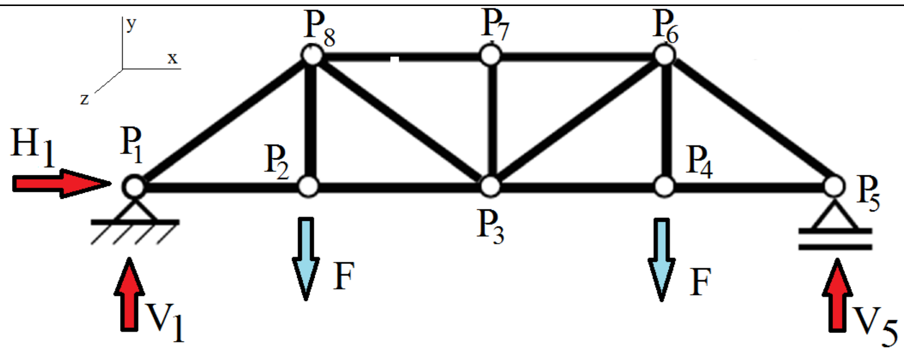


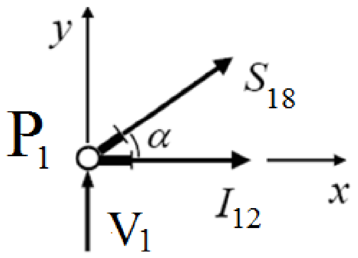
Total Shear Force

Solution (B6) → Total Shear Force → Graph → Animation.



F. ANALYSIS OF RESULTS

F.1 / Analysis of the results obtained by FEM
It is observed that, despite the fact that the modeling of the beam structure was performed with the help of 1D bodies, the results obtained are suggestive, being presented in a 3D environment. From the point of view of the total deformations, it is observed that the maximum value is 0.5 mm in the middle area of the metal structure. In the Ox direction, the maximum displacement is obtained in the bearing corresponding to point P5, having a relatively small value, 0.2 mm. It is observed that the areas with high shear efforts are those corresponding to the assembly points of the sections and those required for bending being the middle areas of the sections (explained by the maximum value of the forces arms in the nodes). Examining the graphical representation of the axial forces, it is observed that the sections located in the lower part of the structure (segments 1-2, 2-3, 3-4, 4-5) are subject to stretching stress - represented in red and those located in the upper part of the structure (1-8, 8-7, 7-6, 6-5) are required for compression - represented by the color blue. The sections in the middle of the structure (8-3, 7-3, 6-3) are very little stressed axially, the value of the efforts tending towards 0. The information regarding the deformations, corroborated with the information regarding the internal stresses, the combined maximum stresses lead to the conclusion that the structure withstands loads without problems, the values of maximum stresses not exceeding 7 x 10^6 Pa, value well below the allowed material limit.
F.2 Prezentarea rezultatelor obţinute prin metoda clasică / Presentation of the results obtained by the classical method
Number the nodes: N = 8. The number of beams b = 13 is established, as well as the number of simple external connections r = 3. The static determination condition is checked: 2N = b + r.

Calculate the reactions in the bearings corresponding to points P1 and P5 using the solidification method, writing the equilibrium equations.
$\sum M_{(P1)} = -F \cdot 4a - F \cdot 12a + V_5 \cdot 16a = 0$ $\sum M_{(P5)} = -F \cdot 4a - F \cdot 12a + V_1 \cdot 16a = 0$ $\sum F_{(Oy)} = V_1 + V_5 - 2F = 0$ We obtain the results: $V_1 = V_5 = F = 5.000$ N; $H_1 = 0$. Isolate node P1 and represent it graphically, writing the equilibrium equations.



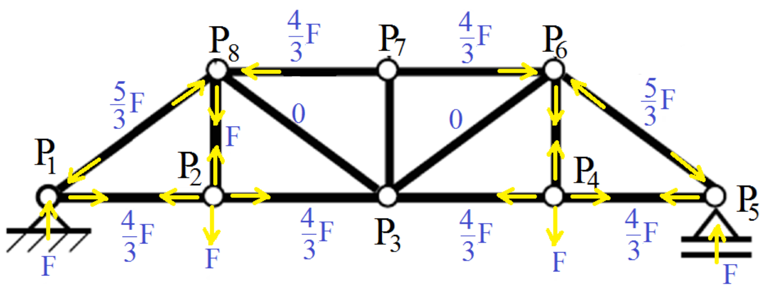
$$\sum F_{(Ox)} = I_{12} + S_{18} cos\alpha = 0$$
$$\sum F_{(Oy)} = F + S_{18} sin\alpha = 0$$

Given that, according to the imposed geometry of the beam structure, $\cos\alpha$ = 4/5 and $\sin\alpha$ = 3/5, the following values are obtained for the axial stresses in sections 1-2 and 1-8, depending on the force F :

I_{12} = 4/3 F = 6.666 N;

S_{18} = -5/3 F = -8.333 N

In the same way, the other axial stresses in each section of the beam structure will be calculated. The following distribution of efforts will be obtained:



F.3 Comparative analysis of results

Using classical methods of *Strength of Materials*, the results are obtained by relatively simple calculations but which require significant time resources, which are directly proportional to the level of complexity of the analyzed structure.

G. CONCLUSIONS

Several sub-chapters of the analysis can be addressed in this subchapter.

From the point of view of the pre-processing phase, it can be seen that the use of 1D bodies involves minimal resources for both modeling and discretization. Another strong point is that the transverse profile of the sections can be modified / oriented very easily, without influencing the basic shape of the beam structure. Moreover, it is possible to use different profiles for each section. The sections can be connected in several ways, depending on the central axis of the profiles used.

The introduction of supports, constraints and demands is quick and easy. The declaration of the materials, as well as the discretization of the beam structure are controllable processes, which can be done automatically or manually.

Comparing the results obtained by the classical method and FEM, it can be seen that they are comparable, at least in the case of axial stresses, a case that was calculated classically, the finite element method providing much more data, over time and with much resource consumption. smaller.

It can be seen that the structure of the beams is very little required, and much smaller profiles can be used in order to achieve savings. Changing the profile of the beam sections and recalculating is done in a very short time, being an easy procedure.

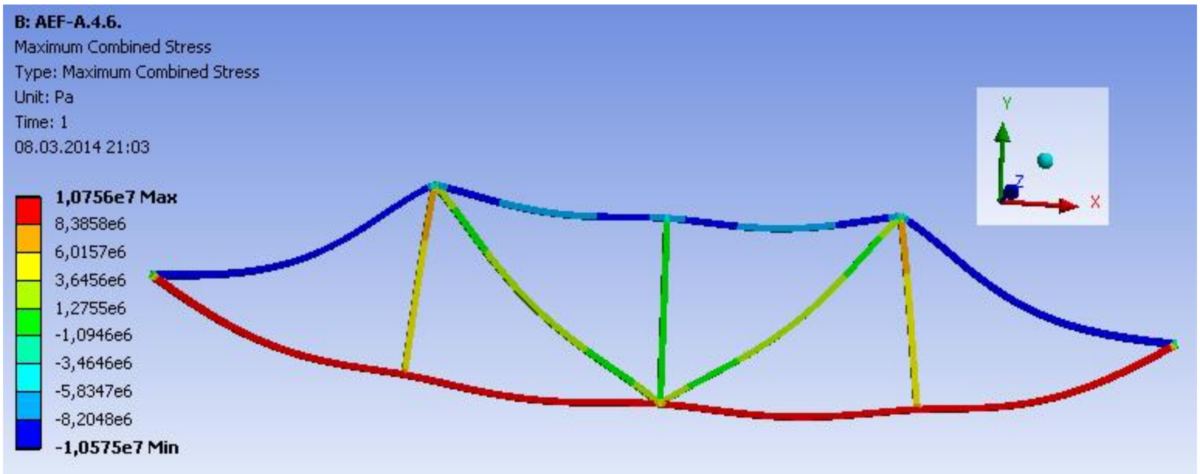
For example, the 80 x 80 x 5 mm rectangular pipe profile will be replaced with 50 x 50 x 5. The result of *Maximum Combined Stress* is shown below.

DM → Modeling → Concept → Cross Section → Rectangular Tube → Details View → **Details of Rect Tube 2** → Sketch: Teava_rect_50x50x5; **Dimensions:** W1 = 50 mm, W2 = 50 mm, t1 = 5 mm, t2 = 5 mm, t3 = 5 mm, t4 = 5 mm → Generate.

DM → ↲ Line Body → Details View → **Details of Line Body** → Cross Section: Teava_rect_50x50x5;

M → ↲ → Line Body → Update Selected Parts → Update: Use Geometry Parameter Value → Solve.

As the section decreases, the values of the combined stresses and strains increase, but do not reach the maximum permissible values.



Application: FEA-A.15
Self-induced vibration modes

KEY WORDS
Linear static analysis, modal analysis, eigenfrequencies, eigenmodes, eigenmodes, planar geometric model, linear material, 1D finite element, linear finite element

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction
In many practical situations for the design of complex mechanical systems it is necessary to know the own frequencies and modes of vibration of some components or even of the whole. These parameters, invariable with time, determined in the conditions of observing the equilibrium configuration, are intimate characteristics of the analyzed structure depending on the shape, dimensions and material. The determination of the frequencies and of the own vibration nodes of the mechanical components or structures can be done by means of the modal analysis. Natural frequencies and vibration modes are very important parameters for the design phase because they provide information about the dynamic behavior of the analyzed structures. The modal analysis within the ANSYS program is a linear analysis. Any nonlinearity such as plasticity and contact elements is ignored, even if it is defined. Modal analysis is used to calculate the natural frequencies and modes of deformation of the structure.
A.2 Application description
Bridges made of lattice beams are characterized by high rigidity, being generally made of elements made of steel. Identifying your own modes and frequencies of vibration is particularly important to take into account in the design process the values of the frequencies of certain demands: natural (earthquakes, strong winds) or artificial (induction of vibrations by vehicles crossing the bridge) to avoid total or partial destruction of the structure. At the same time, because tensions can occur in the structure at temperature variations, the bridge is fixed at one end by means of a rotating joint with a bolt and at the other end it is supported and guided allowing translation.
A.3 Application goal
The purpose of this application is to identify its own modes and frequencies of vibration for the bridge-type structure of lattice beams in order to avoid its resonance phenomena. The displacement fields for each vibration mode will be

presented in order to optimize the construction of the structure, respectively to minimize its weight in compliance with the deformation and strength restrictions.

B. PREPARATION OF THE MODEL FOR ANALYSIS

B.1 The model definition

The modal solution is obtained following a modal analysis which consists in completing the following steps:

- model construction;
- applying loads and obtaining the solution through structural analysis;
- expanding modes;
- viewing the results.

In order to draw up the finite element analysis model associated with the above application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The basic equation solved in a typical unamortized modal analysis typical for the ANSYS program is given by the classical problem of eigenvalues:

$$[K] = \omega i^2 [M]$$

where [K] is the stiffness matrix; is the shape vector (eigenvector) of mode i;
ω_i is the natural frequency of mode i (ω_i^2 is the eigenvalue); [M] is the mass matrix.
Among the methods for solving this equation, recommended in the ANSYS program, the Lanczos vectors method will be used in this paper. The static stresses of the bridge-type mechanical structures can be overlapped by the dynamic stresses which, together with the static ones, can cause the destruction of this part. One of the dynamic stresses to which a bridge is subjected is the stress due to vibrations caused by various causes during use (passing of people, vehicles, vibrations due to machinery or work equipment, weather stresses - strong wind, etc.).
The mechanical structure studied in this paper is considered independent, without mechanical connections and other constraints. This method of calculation was approached because the modeling of related elements would lead to large dimensions of finite element models, which would have a negative effect on the accuracy of the results. Thus, the modal analysis of the mechanical structure will be performed in order to obtain indications on the occurrence of the resonance phenomenon.

B.3 Characteristics of the material
For finite element analysis the strength characteristics of the material, S235 steel (equivalent to OL 37) are: • modulus of longitudinal elasticity, E = 204,000 N / mm^2; • transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

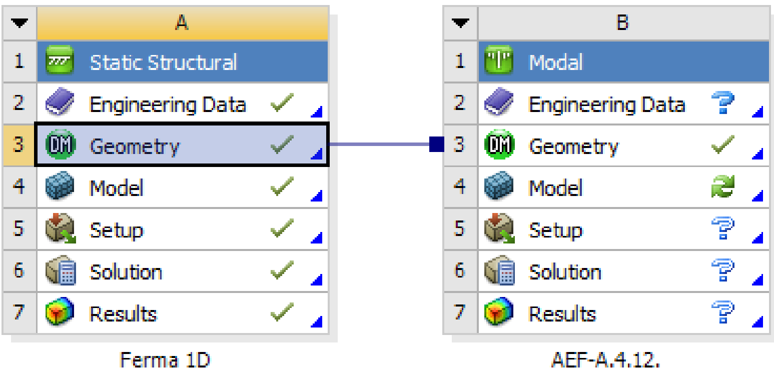
Creating the project

The drawing made in the previous application "*Linear static analysis of bar structures*" will be taken over.

In order to take over a geometry previously made in another analysis, the following commands will be executed, in the order presented:

WB → Open... (navigate to Windows Explorer and select the previously made "Farm 1D" file) → ↵↵ Modal → will get a window with two analysis structures (Static Structural and Modal).

By holding down Geometry on the command in the Static Structural analysis structure, it is dragged over the command Geometry in the Modal structure until it turns into Share B3, when the mouse key is released OK. The link between the two projects is shown in the figure below → Save As... → File name: *Modal_ex* → Save.

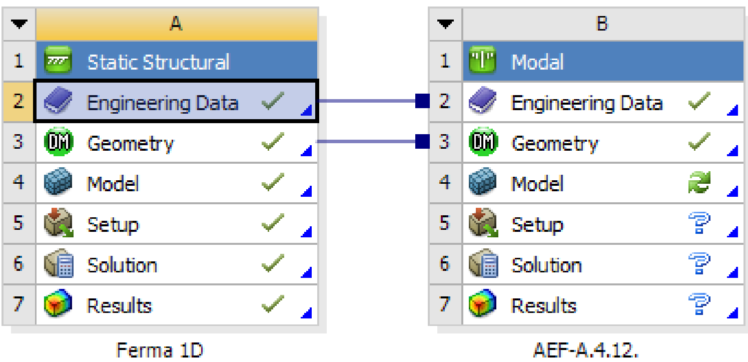


Problem type setting (3D)

A: ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, Advanced Geometry Options, ↵ Analysis Type, [select from list ↵, ↵3D] → [close window ↵ X].

C.2 Modelling of material characteristics

The material of the new structure can be introduced by taking over the characteristics from the main window, with the "pull" procedure from *Static Structural*.



Changing the value of the modulus of elasticity can be done following the commands:

→ ↳ Engineering Data → Edit... → Properties of Outline Row 3: Structural Steel [the value of the longitudinal modulus of elasticity changes , Young's Modulus = 2,04 x 10^{11} Pa] → Return to Project.

Properties of Outline Row 3: Structural Steel

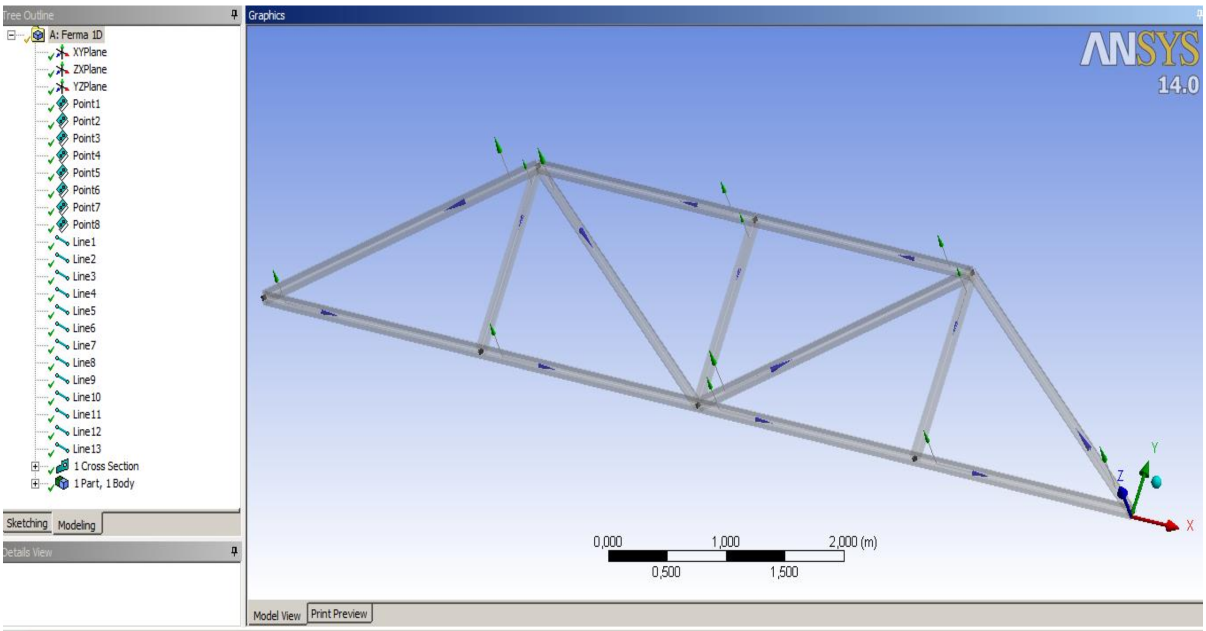
	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	Alternating Stress Mean Stress	Shear Modulus abular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

If the window Properties of Outline Row 3: Structural Steel is not visible, the *Outline* and *Properties* options will be activated in the *View* menu.

C.3 Creating the geometric model using DesignModeler (DM)

Making the beam structure with lattice

→ ↲↲ Geometry from the *Static Structural* analysis structure the geometric model of the lattice beam will open.



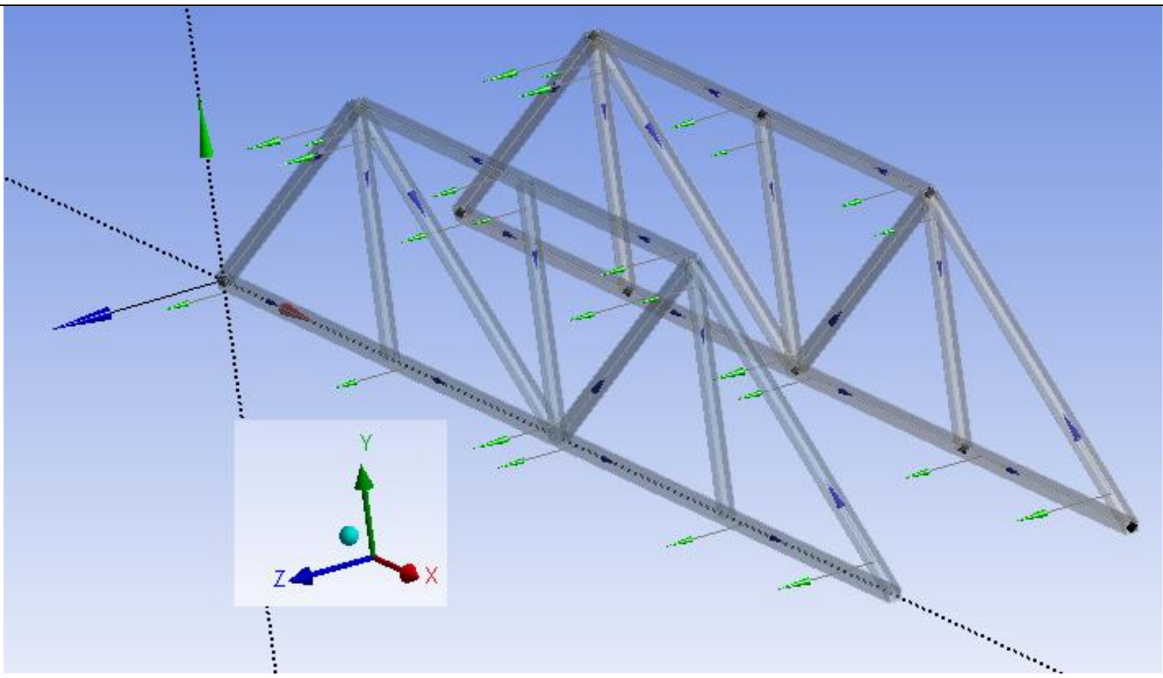
The geometry of the bridge can be continued in two ways: either by building a number of 8 points corresponding to the beam in a plane parallel to the one in which the original model was built (at z = 1.5 m) or by copying the existing beam at distance of 1.5 m on the Oz axis.

Draw a plane parallel to xOy, at a distance of 1.5m.

→ ↲ New Plane → Details View → **Details of Plane 4** → Type: From Plane → Base Plane: XY Plane → Transform 1 (RMB): Offset Z → FD1, Value 1: 1,5 m → Generate.

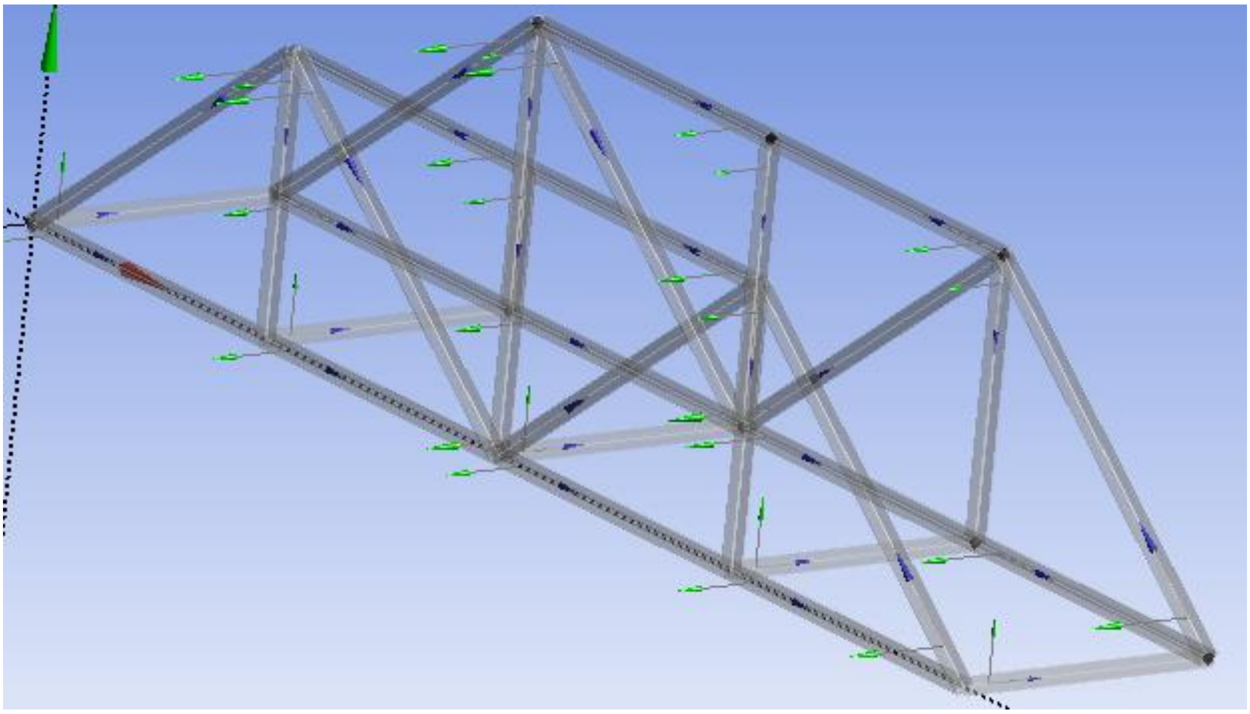
Copy the existing beam into the newly created plane.

→ Create → Body Operation → Details View → **Details of BodyOp1** → Type: Move → Bodies (select the lattice beam using selection filter) → Preserve Bodies?: Yes → Source Plane: XY Plane → Destination Plane: Plane 4 → Generate.



When there are two parallel lattice beams, the points on the lower base will be joined.

→ Concept → Lines From Points → the corresponding points are joined → Generate.



Assigning a transverse profile to the metal structure

In terms of section geometry, the newly created segments will have the same properties as the original beam. The beam imported from the previous analysis is assigned a profile, the procedure is as follows:

→ ↵ Line Body → Details View → **Details of Line Body** → Cross Section: Teava_rect_80x80x5;

Modeling → View → Cross Section Solids [the 3D section view option is activated] → check that the profiles are oriented correctly; the profile orientation is displayed using green arrows. If the profile is not symmetrical and is not project oriented, the orientation can be changed accordingly: ↵ Line Body → Details View → **Line Body-Edge** → Reverse Orientation?: Yes] → Offset Type: Centroid → Generate.

Saving the geometric model

→ (Save Project) → File → Close Design Modeler.

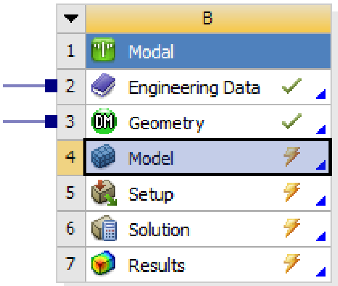
→ ↳ Geometry → Properties → Properties of Schematic A3: Geometry → **Advanced Geometry Options** → Analysis Type: 3D → Save.

C.4 Finite element modelling

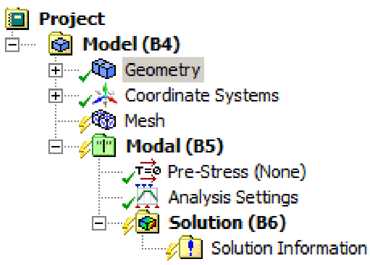
C.4.1 Activate the discretization module and set the finite element type

→ ↵↵ Model → is launched from the analysis structure *Modal, the* modul *Mechanical [ANSYS Multiphysics]*.

↵ Geometry → Details of "Geometry" → **Definition** → Element Control: Program Controlled .



AEF-A.4.12.



Details of "Geometry"

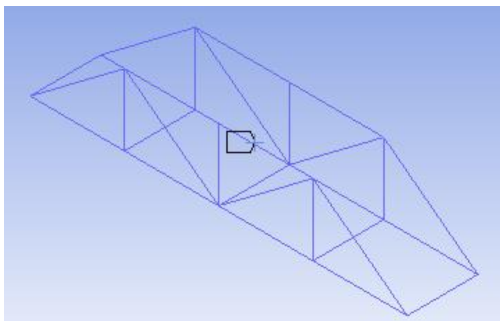
Definition	
Source	C:\Users\BS\Dropbox\Pod 1D vibratii proprii\Pod 1D - ve...
Type	DesignModeler
Length Unit	Meters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Properties	
Statistics	
Basic Geometry Options	
Advanced Geometry Options	

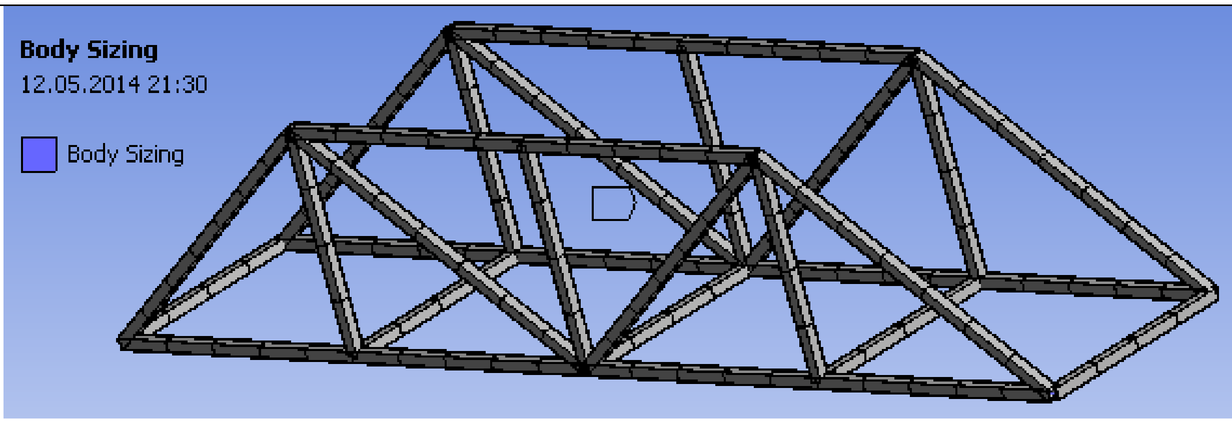
C.4.2 Model discretization and finite element size setting

→ ↵ Mesh → Mesh Control → Sizing → Details of "Body Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ structure geometry, using the selection filter (Body)] Apply; **Definition** → Element Size: Default → Update

Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	Default
Behavior	Soft





C.5 Supports and restraints modelling

Input restraint

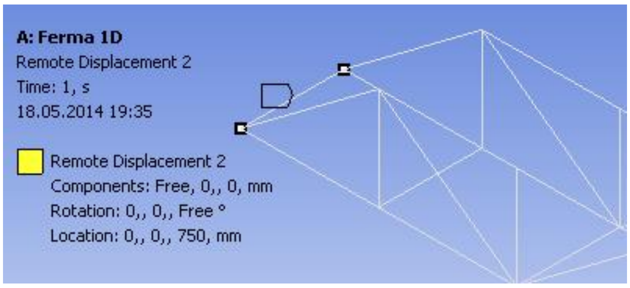
→ ↲ ? Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" →

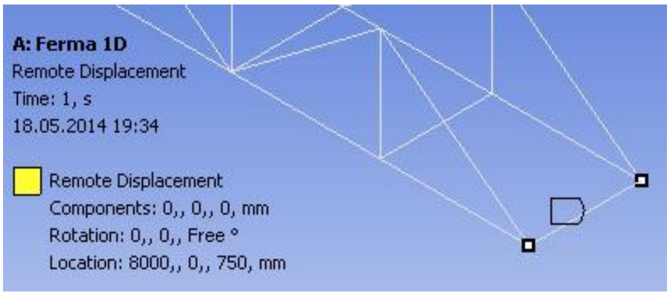
Scope → Geometry: [will select, with ↲, holding down the Ctrl key, points P1 and P1', using the selection filter (Vertex)] → Apply; **Definition** → X Component: 0, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.

Will repeat the actions for points P5 şi P5':

→ ↲ ? Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" →

Scope → Geometry: [will select, with ↲, holding down the Ctrl key, points P5 and P5', using the selection filter (Vertex)] → Apply; **Definition** → X Component: Free, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.

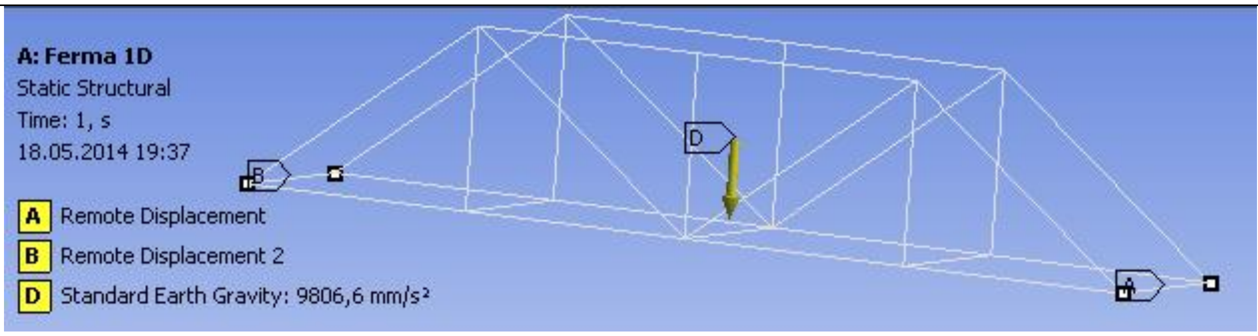




C.6 Load modeling

Introduction of gravitational acceleration

→ ↲ ? Static Structural (B5) → Inertial → Standard Earth Gravity (the selection of the gravitational acceleration implies the taking into account of the own weight of the metallic structure) → Details of "Standard Earth Gravity" → **Definition** → Direction: -Y Direction.



D. SOLVING THE FEA MODEL

D.1 Select the types of results

In order to select the final data types to be analyzed after the launch of the calculation module, the series of commands presented below will be followed.

→ ↳ Solution (B6) → <u>Insert</u> → Deformation → Total [use the commands in the command box open with ↳].

The same result can be obtained by using commands:

↲ Solution (B6) → Deformation → Total [the buttons in the menu bars are used]

In order to obtain suggestive results, analyzes will be performed with several types of profiles of the beams of the metal structure: rectangular pipes with dimensions 80 x 80 x 5 mm, 60 x 60 x 5 mm, 50 x 50 x 5 mm and profiles I of sections of 3800 mm^2, 950 mm^2, 237.5 mm^2.

D.2 *Launching the solving module*

→ ↲ Solution (B6) → Solve.

Solution (B6)
Solution Information
Total Deformation
→ Solve →
Solution (B6)
Solution Information
Total Deformation

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu bars:

Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

<u>Total deformation view</u>

↲ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

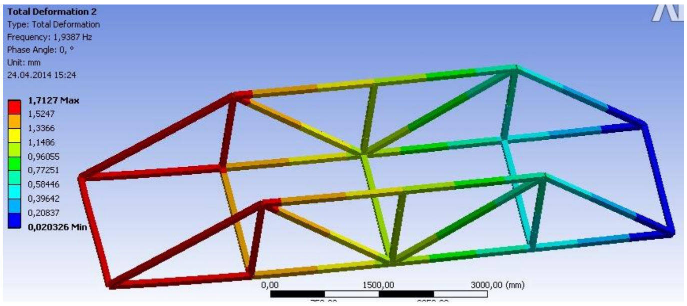
Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation. The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command .

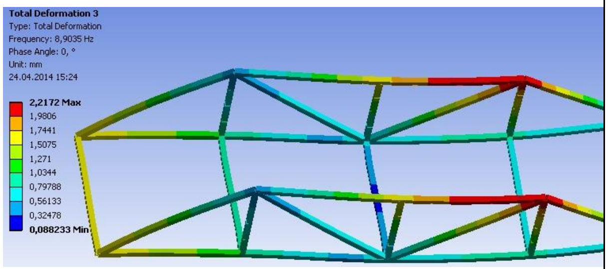
The following are some results of the values of eigenfrequencies and vibration modes for the various profiles analyzed.

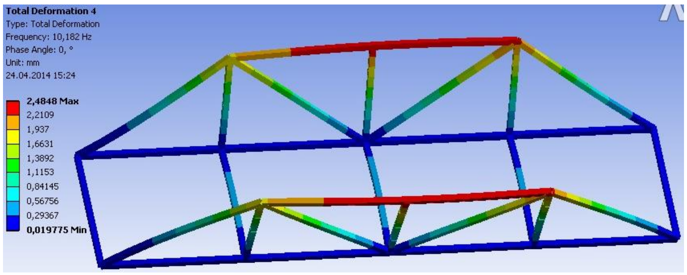
Rectangular pipe 80 x 80 x5 mm

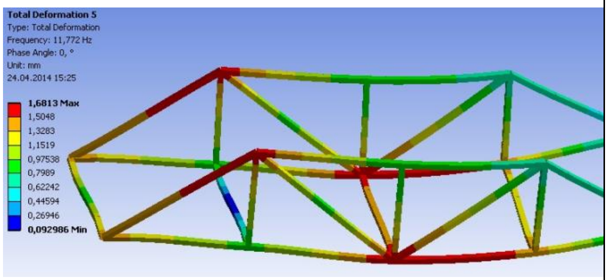
Volume	9,45e+007 mm³
Mass	741,82 kg
Length	63000 mm
Cross Section	Teavarectangulara80x80x5
Cross Section Area	1500, mm²

	Mode	Damped Frequency [Hz]
1	1,	2,1807e-005
2	2,	2,932e-005
3	3,	1,9387
4	4,	8,9035
5	5,	10,182
6	6,	11,772





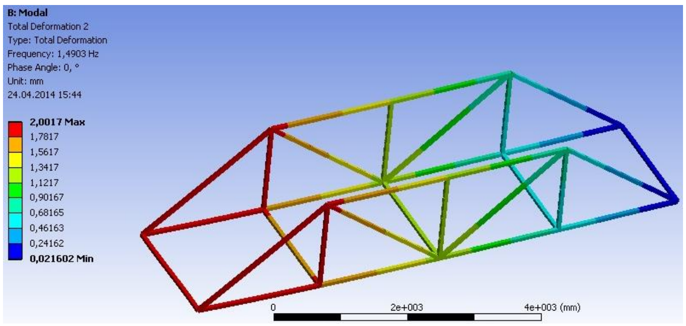


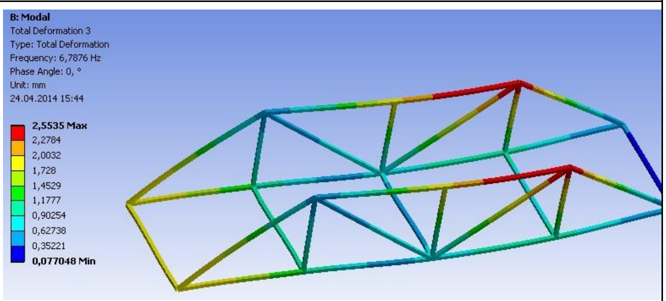


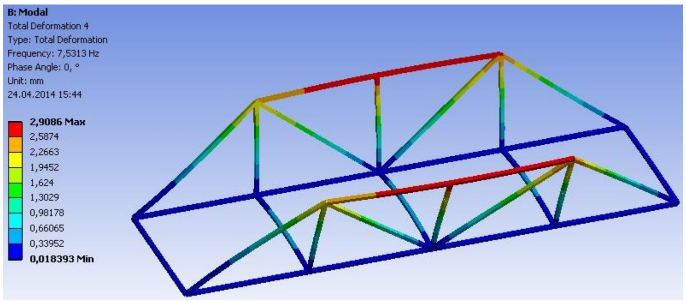
Rectangular pipe 60 x 60 x 5 mm

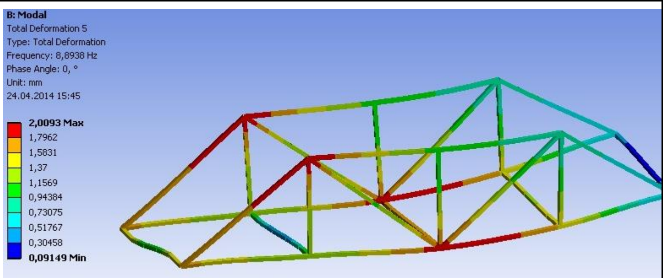
Volume	6,93e+007 mm³
Mass	544,01 kg
Length	63000 mm
Cross Section	Teavarectangulara60x60x5
Cross Section Area	1100, mm²

	Mode	Damped Frequency [Hz]
1	1,	1,9005e-005
2	2,	2,9813e-005
3	3,	1,4903
4	4,	6,7876
5	5,	7,5313
6	6,	8,8938





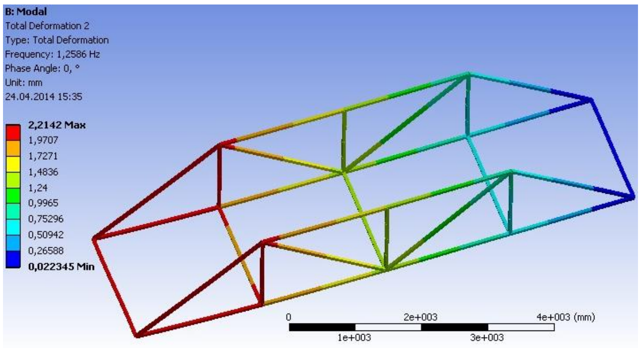


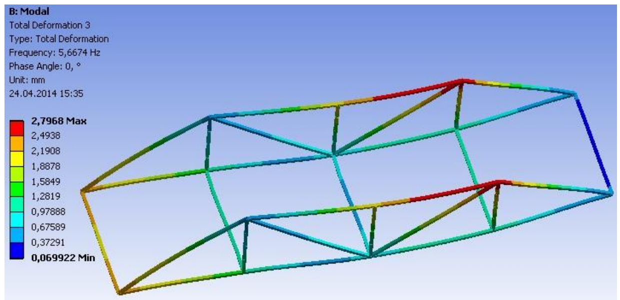


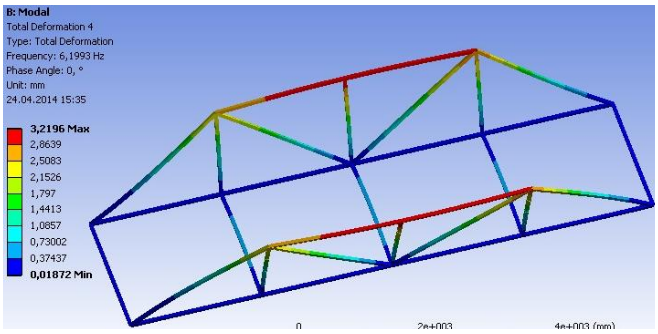
Rectangular pipe 50 x 50 x 5 mm

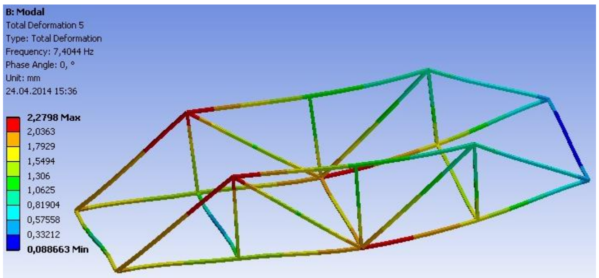
Volume	5,67e+007 mm³
Mass	445,1 kg
Length	63000 mm
Cross Section	Teavarectangulara50x50x5
Cross Section Area	900, mm²

	Mode	Damped Frequency [Hz]
1	1,	7,3932e-006
2	2,	2,6739e-005
3	3,	1,2586
4	4,	5,6674
5	5,	6,1993
6	6,	7,4044





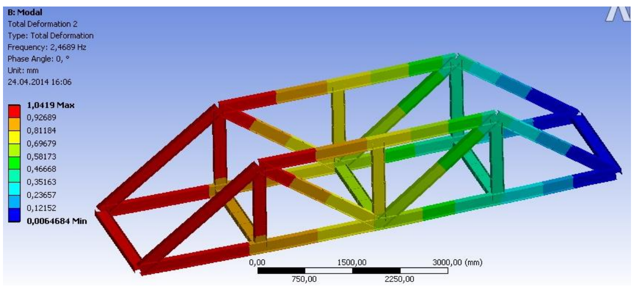


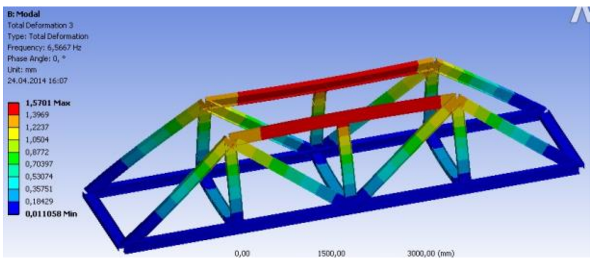


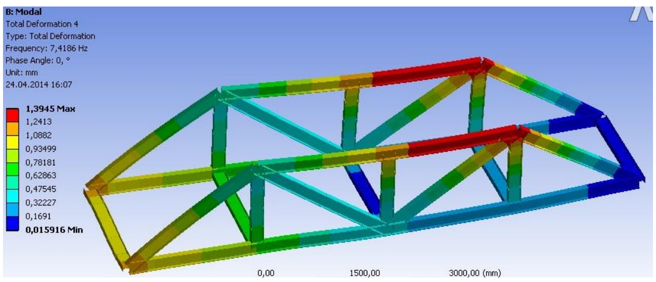
Profil I1, secțiune 3800 mm²

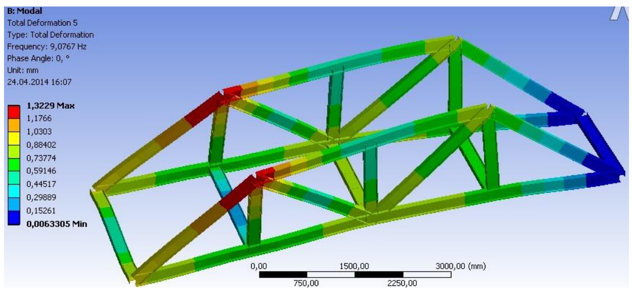
Volume	2,394e+008 mm³
Mass	1879,3 kg
Length	63000 mm
Cross Section	I1
Cross Section Area	3800, mm²

Mode	Damped Frequency [Hz]
1,	0,
2,	0,
3,	2,4689
4,	6,5667
5,	7,4186
6,	9,0767





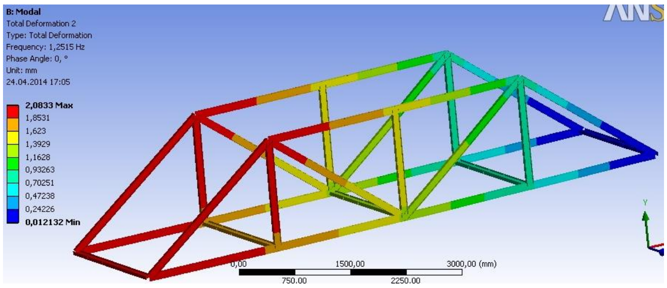


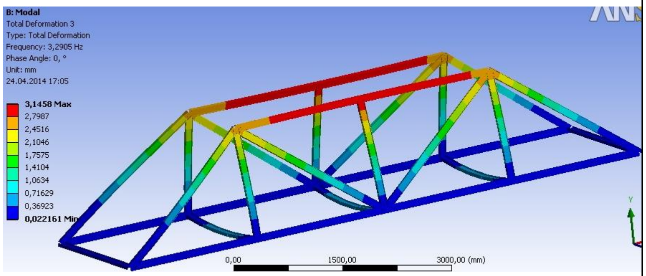


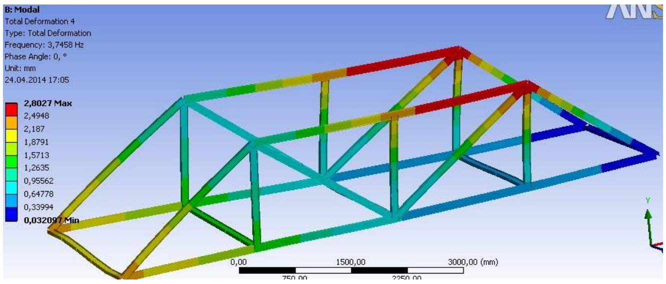
Profil I2, secțiune 950mm²

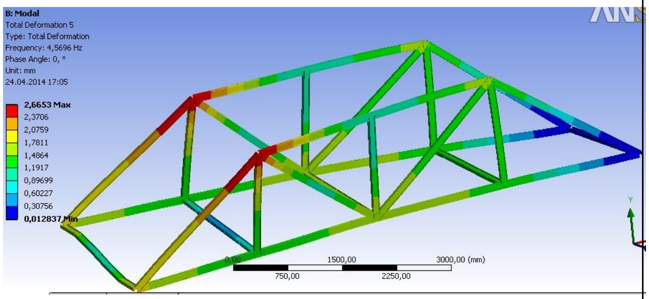
Volume	5,985e+007 mm³
Mass	469,82 kg
Length	63000 mm
Cross Section	I2
Cross Section Area	950, mm²

Mode	Damped Frequency [Hz]
1,	0,
2,	0,
3,	1,2515
4,	3,2905
5,	3,7458
6,	4,5696





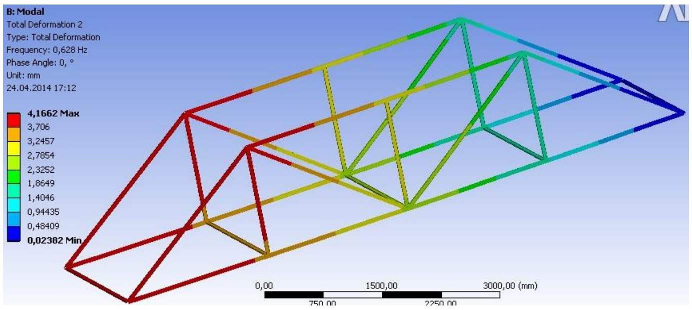


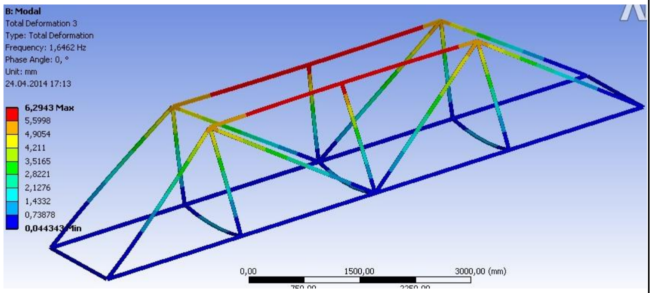


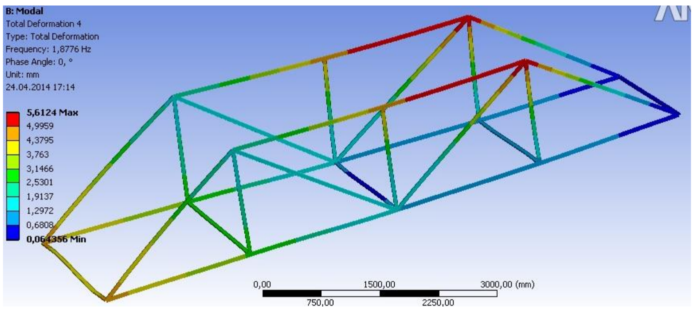
Profil I3, secțiune 237,5 mm²

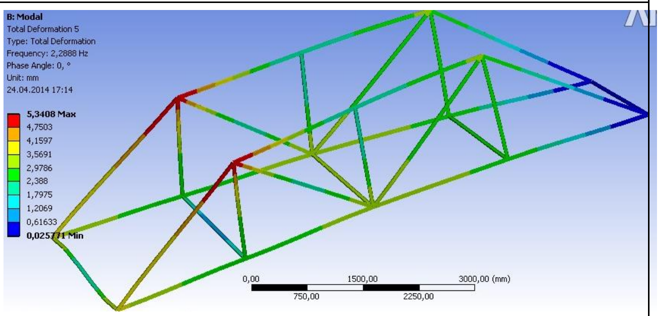
Volume	1,4962e+007 mm³
Mass	117,46 kg
Length	63000 mm
Cross Section	I3
Cross Section Area	237,5 mm²

Mode	Damped Frequency [Hz]
1,	0,
2,	0,
3,	0,628
4,	1,6462
5,	1,8776
6,	2,2888





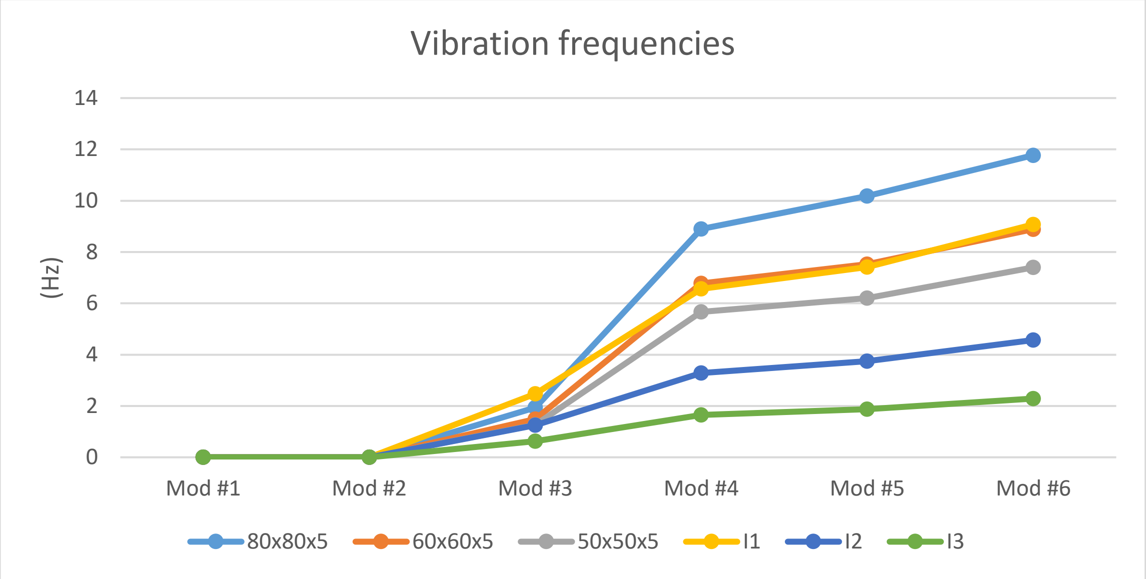




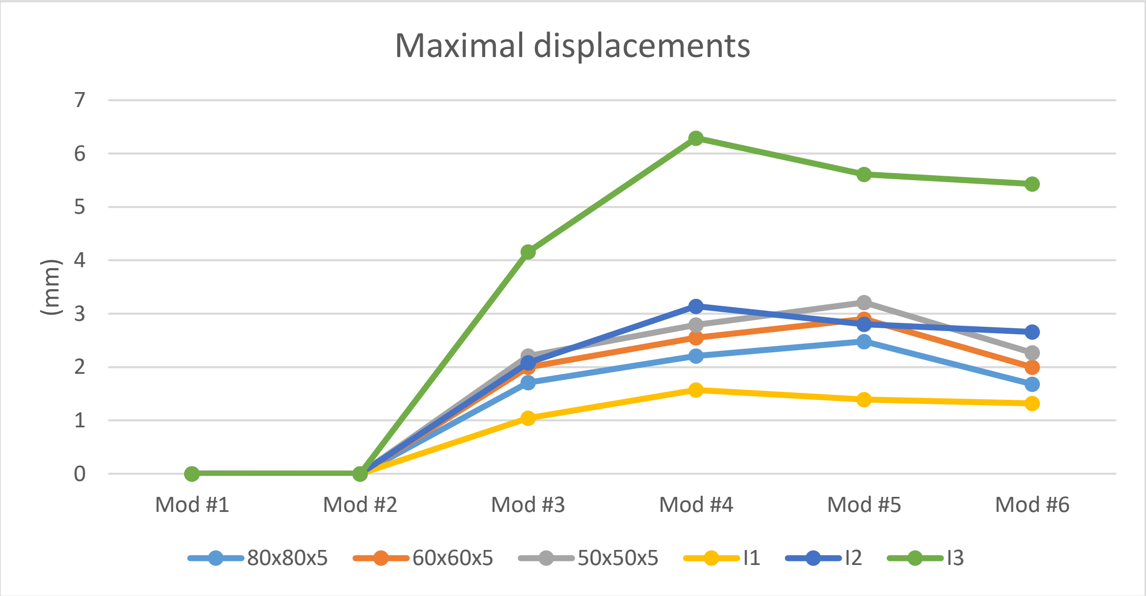
F. ANALYSIS OF RESULTS

It is observed that, despite the fact that the modeling of the bar structure was performed with the help of 1D bodies, the results obtained are suggestive, being presented in a 3D environment.

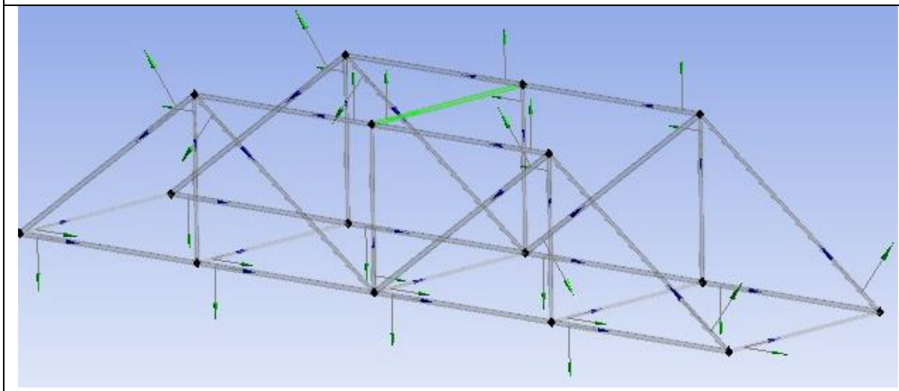
From the point of view of the recorded own frequencies, it can be concluded that with the increase of the beams section, the value of the frequencies will increase, regardless of the transversal profile used. This is observed for both types of profiles analyzed: rectangular profile and for profile I.



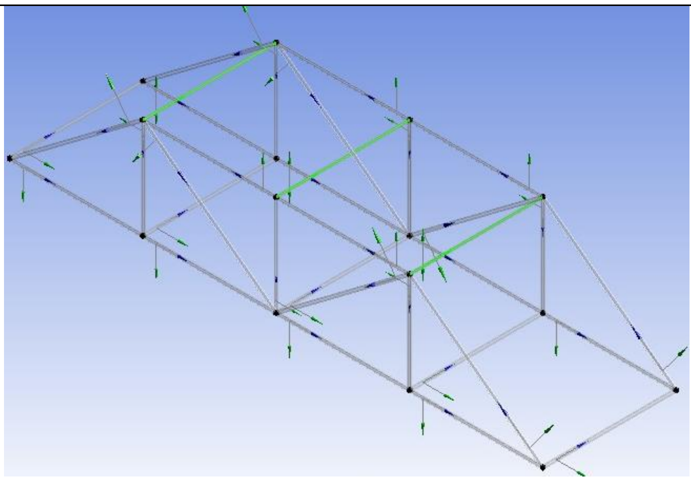
For equivalent profiles in terms of the value of the section surfaces, it can be seen that the rectangular profile generates its own vibrations with higher frequencies than the I profile.



From the point of view of the total displacements, it is observed that the maximum values are found in the own vibration modes Mode # 4 or Mode # 5, after which they decrease with the increase of the own vibration frequencies. Some applications aim to increase the rigidity of a structure or change its own frequencies, based on an existing structure. In this case, for the structure built from profile I3 (with the smallest cross section of the analyzed ones) two modifications are considered: the installation of some sleepers (case 1 - a sleeper, case 2 - 3 sleepers) in the upper part, as follows . It is observed after the analysis of the vibration modes that the values of the own frequencies have changed compared to the original structure in the direction of the increase.



Mode	Damped Frequency [Hz]
1,	1,027e-005
2,	0,
3,	0,63347
4,	1,9132
5,	2,4412
6,	2,7407



	Mode	Damped Frequency [Hz]
1	1,	0,
2	2,	0,
3	3,	0,66262
4	4,	1,9957
5	5,	2,5659
6	6,	4,0492

G. CONCLUSIONS

From the point of view of the pre-processing phase, it can be seen that the use of 1D bodies involves minimal resources for both modeling and discretization. Another strong point is that the transverse profile of the sections can be modified / oriented very easily, without influencing the basic shape of the bar structure. Moreover, it is possible to use different profiles for each section. The sections can be connected in several ways, depending on the central axis of the profiles used.

The introduction of supports, constraints and demands is quick and easy. The declaration of the materials, as well as the discretization of the bar structure are controllable processes, which can be done automatically or manually.

The modal analysis of a lattice beam structure is a relatively simple activity, and various modifications of the structure can be made depending on the objectives pursued. Changing the profile of the beam sections and recalculating is done in a very short time.

Application: FEA-A.16
Static analysis of bar mechanisms

KEY WORDS
Linear Static Analysis, Plane Geometric Model, Plane Voltage State, Linear Material, 1D Finite Element, Linear Finite Element, Machine Element, Mechanical Subassembly, Bar Mechanisms, Joints

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

In various finite element analysis (FEA) applications it is necessary to model not only a single part, but also a whole mechanism, including the joints between its elements.

Complex plane or spatial mechanisms can be reduced to bar mechanisms (levers) and toothed mechanisms (gears and racks). The methods of studying the complex mechanisms with bars and gears are very diverse, especially in the field of kinematics, starting from the hypothesis of the rigidity of the components. This paper aims to study the behavior of the elements of a bar mechanism, taking into account their elastic behavior.

A.2 Application description

From a constructive point of view, the jack in the figure below is a flat mechanism consisting of four bars of various sections mounted on a support, a screw and a nut that make a helical coupling. The analyzed jack can operate within two positions: A - start lifting and B - maximum lifting.

The vertical movement of the upper plate 5 is determined by the change of the positions of the side segments 2, mounted by means of support joints 1, due to the axial movement of the nut 6. The screw 3 is actuated by means of the crank 4.

All the joints between the segments 2 and the support 1, between the segments 2 and the upper plate 5, as well as those between the lower and the upper segments are flat rotational couplings.

By rotating the crank 4, thanks to the nut 6, the side segments of the jack tend to approach each other, generating vertical movement, thus lifting the vehicle.

A.3 Application goal
The application aims to determine the maximum values of the fields of displacements, deformations, internal stresses produced in operation on the component elements. For this analysis, the use of one-dimensional elements was considered due to the simplicity of the geometric construction, the ease of modifying the profile of the studied elements but also the main objective - to use the joints in the study with finite elements.

B. PREPARATION OF THE MODEL FOR ANALYSIS

B.1 The model definition

In order to draw up the finite element analysis model associated with the present application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

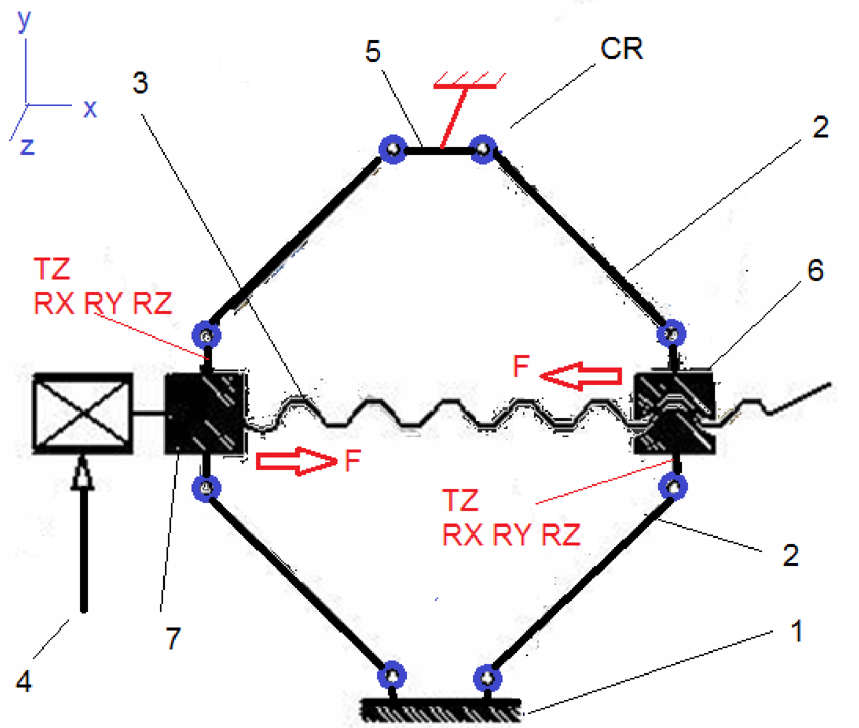
B.2 The analysis model description

The dimensions of the studied mechanism, respectively the lengths of the segments and their sections, are presented in chapter C.3.2, C.3.3 and C.3.4, these being, on the one hand, taken from the specialized literature and, on the other hand, imposed from constructively so that the problem is determined.

The construction of the segments 2, of the vertical zones afferent to the guide bush (7) and the nut 6, the upper plate 5 are constructed in the form of one-dimensional bars.

The connections between these bars are made with simple rotating joints (CR in the adjacent drawing). In addition, the vertical side segments will have translational movements only in the plane of the mechanism, without rotations.

The external loads, generated by the mass of the raised vehicle, are shaped by the rigid fixing of the support 1 and the upper plate 5, and the forces generated by the screw act on the horizontal axis, on the lateral vertical segments, in the direction of approaching the lateral segments.



B.3 Characteristics of the material

For finite element analysis the strength characteristics of the material, S235 steel (equivalent to OL 37) are:

- modulus of longitudinal elasticity, E = 204,000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Create of the project

WB, Toolbox: ↲ ⊟ Analysis Systems → ↲↲ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural în *Cric auto*].

Problem type setting (3D)

A : ↳ Geometry → ↲Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↲Analysis Type, [select from list ↲, ↲ 3D] → [close window ↲ X].

Saving the project

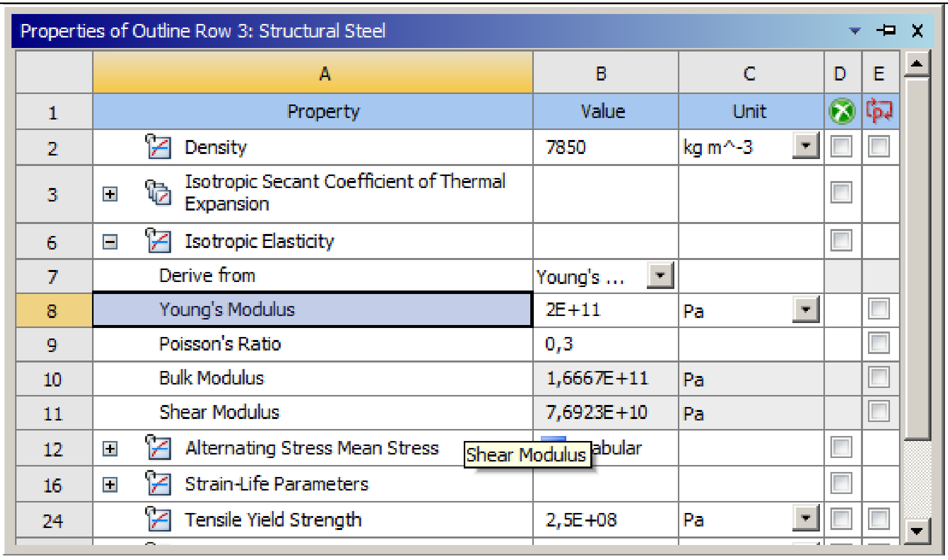
↲ Save As... → WB Save As, File name: [input name, *Cric*] → ↲ Save.

C.2 Modelling of material characteristics

WB, Project Schematic: ↳ Engineering Data ✓ → ↲ Edit... → Outline of Schematic A2: Engineering Data:

↵ Structural Steel , Properties of Outline Row 3: Structural Steel : ⊟ Isotropic Elasticity → Young's Modulus , Young's Modulus , [select from column C (Unit) cu ↵▼, ↵MPa], [input in column B (Unit) value, 204000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

If the window Properties of Outline Row 3: Structural Steel is not visible, the Outline and Properties or Reset Workspace options will be activated in the View menu, then the *Engineering Data Sources* command .


Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	⊞ Isotropic Secant Coefficient of Thermal Expansion				
6	⊟ Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	⊞ Alternating Stress Mean Stress	Shear Modulus abular			
16	⊞ Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

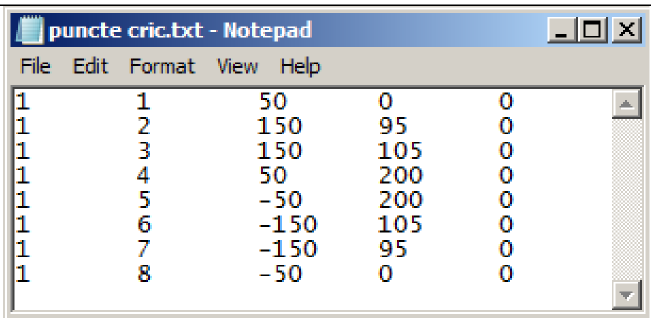
C.3 Geometric modelling

C.3.1 Loading DesignModeler (DM) module

, Project Schematic: ↳ Geometry → ↵ New Geometry... → ANSYS Workbench : ↵ Millimeter , ↵OK.

C.3.2 Generating points

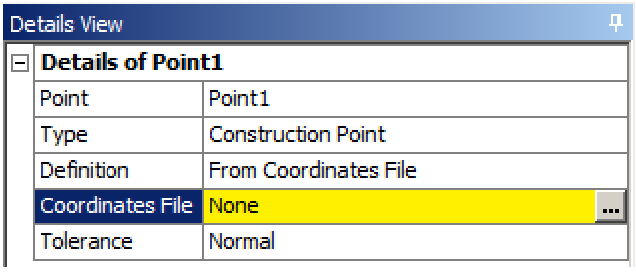
Using *Notepad* program generate a txt file with the form shown next to it, which will be saved as *puncte cric.txt*. It will contain the coordinates of the construction points of the jack.


puncte cric.txt - Notepad

File Edit Format View Help

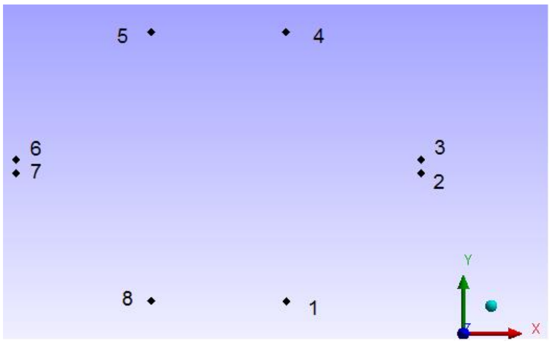
```
1   1   50    0    0
1   2   150   95   0
1   3   150   105  0
1   4   50    200  0
1   5   -50   200  0
1   6   -150  105  0
1   7   -150  95   0
1   8   -50   0    0
```

→ Modeling → Create → Point → Details View → **Details of Point 1** → Definition: From Coordinates File; Coordinates File : select the saved file (*puncte cric.txt*) → Generate


Details View

Details of Point1	
Point	Point1
Type	Construction Point
Definition	From Coordinates File
Coordinates File	None
Tolerance	Normal

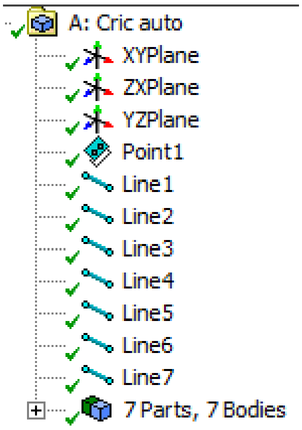
The 8 points in the xOy plane will be obtained automatically, as in the figure below:

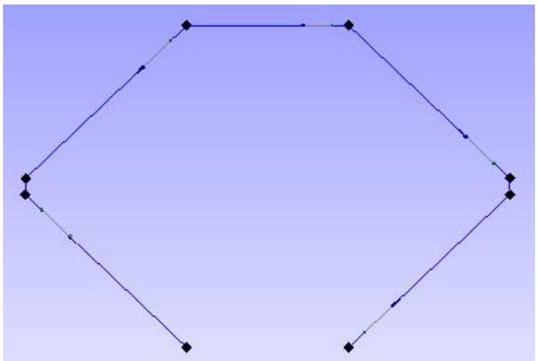


C.3.3 Segments generation

DM → Modeling → Concept → Lines From Points → Details View → **Details of line 1** → Point Segments [while holding down the *Ctrl* key, select points P1 and P2]: Apply → Generate.

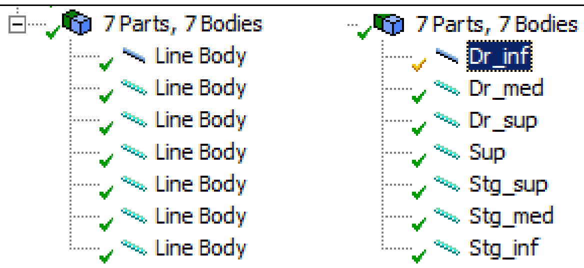
In the same way, using the option *Operation: Add Frozen* din Details View, the segments corresponding to the pairs of points are constructed:: (P2, P3), (P3, P4), (P4, P5), (P5, P6), (P6, P7), (P7, P8). The structure in the figure below will be obtained.





Rename segments

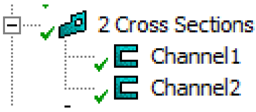
DM → Modeling → Line Body → Rename → the segments created according to the adjacent structure are renamed.

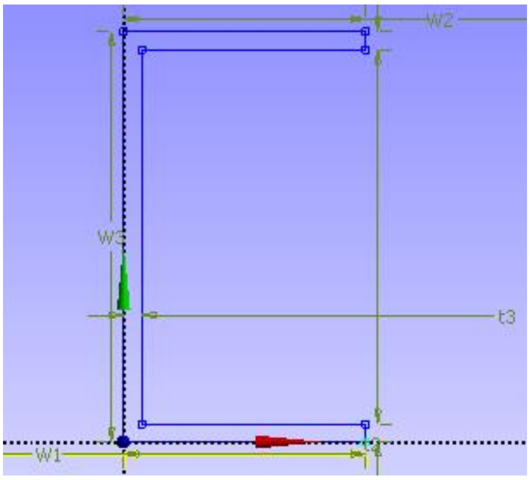


C.3.4 Generation of cross sections

→ Modeling → Concept → Cross Section → Channel Section → Details View → **Details of Channel 1** → Sketch: *Channel1* (U34); **Dimensions:** W1 = 20 mm, W2 = 20 mm, W2 = 34 mm, t1 = 1,5 mm, t2 = 1,5 mm, t3 = 1,5 mm → Generate.

The procedure will be the same for a profile *Channel2* (U30) with dimensions: W1 = 15 mm, W2 = 15 mm, W2 = 30 mm, t1 = 2 mm, t2 = 2 mm, t3 = 2 mm.

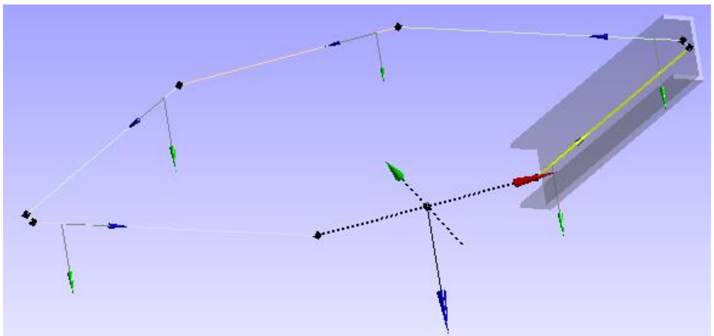


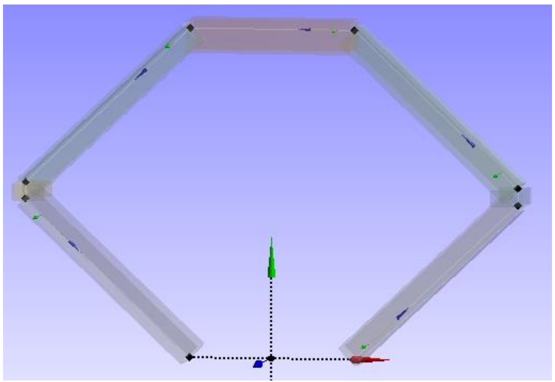


C.3.5 Assigning transverse profiles to the metal structure

In order to complete the procedure for generating a profile, proceed as follows:

→ ↵ Dr_inf → Details View → **Details of Line Body** → Cross Section : Channel2; Offset Type: Centroid; Modeling → View → Cross Section Solids [the 3D section view option is activated] → [check that the profiles are oriented correctly; the profile orientation is visualized with the help of green arrows; if the profile is not symmetrical and is not project oriented, the orientation can be changed accordingly: ↵ Dr_inf → Details View → **Line Body-Edge** → Reverse Orientation? : Yes] → Offset Type : Centroid → Generate.





The last operation is repeated for all segments, taking care that the Channel1 (U34) profile is attached to the *Dr_med, Sup, Stg_med* segments and to the other segments - the Channel2 profile (U30).

C.3.6 Saving the geometric model

→ (Save Project) → File → Close Design Modeler.

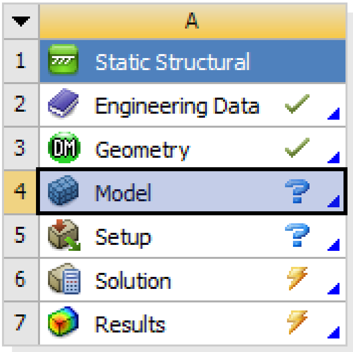
C.4 Finite element modelling

C.4.1 Activate the discretization module and set the finite element type

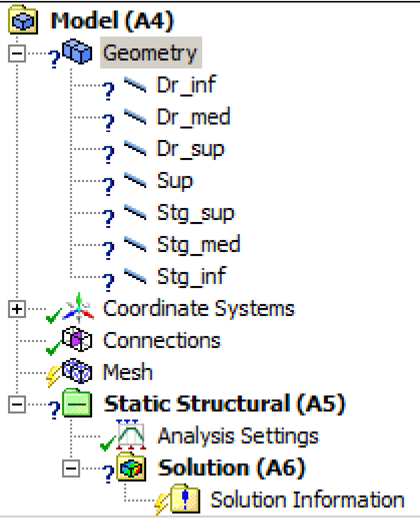
Launch of the finite element modeling module

, Project Schematic ↵↵ Model → launching module *Mechanical [ANSYS Multiphysics]*.

: ↵ Geometry → Details of "Geometry" → **Definition** → Element Control : Program Controlled .



Cric auto



Setting the unit of measurement system

: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

Setting the material characteristics

Outline: ↵ Geometry → Dr_inf → Details of "Dr_inf" → **Material** : ↵ Assignment → [select from list ↵, ↵ Structural Steel (setare implicită / default)]. The operation will be repeated for the other segments as well.

Setting the model type

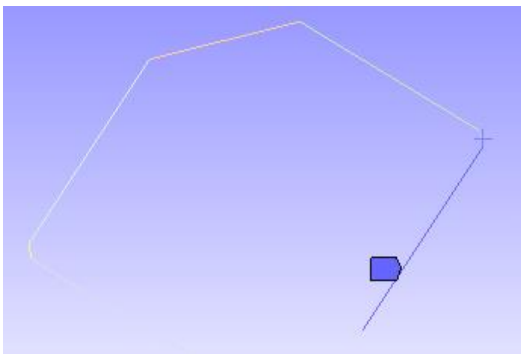
Outline: ↵ Geometry → Details of "Geometry", **Definition** : ↵ 3D Behavior.

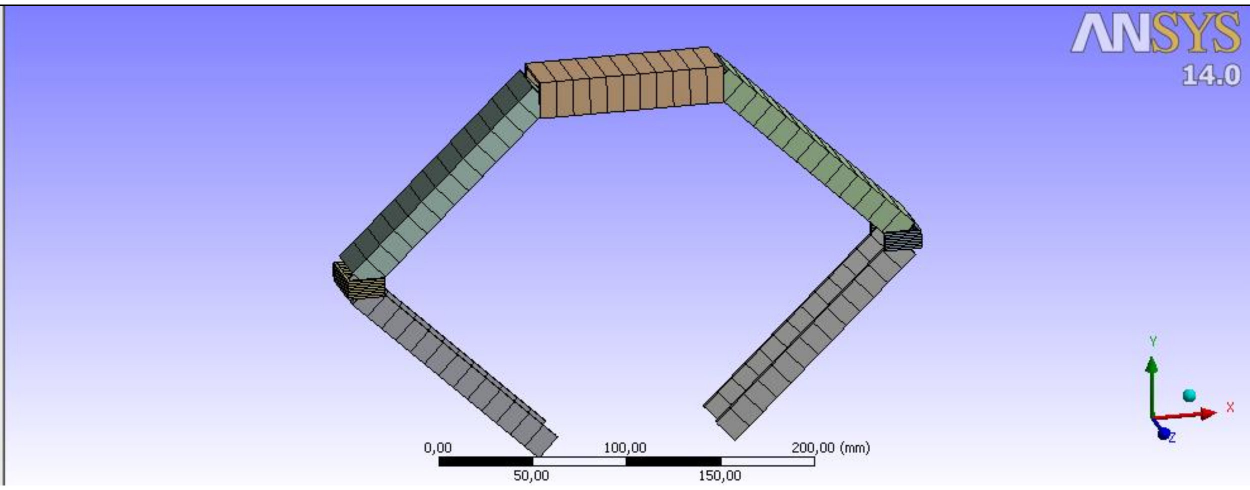
C.4.2 Model discretization and finite element size setting

, Outline: ↵ Mesh → Mesh Control → Sizing → Details of "Edge Sizing" - Sizing → **Scope** → Select Geometry: [a segment of the structure geometry will be selected with ↵ using the selection filter (Edge)] Apply; **Definition Element** → Size: Default → Update. The operation will be repeated for the other segments as well.

Details of "Edge Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Suppressed	No
Type	Element Size
Element Size	Default
Behavior	Soft
Bias Type	No Bias





C.5 Modeling joints and constraints

Introduction of gravitational acceleration

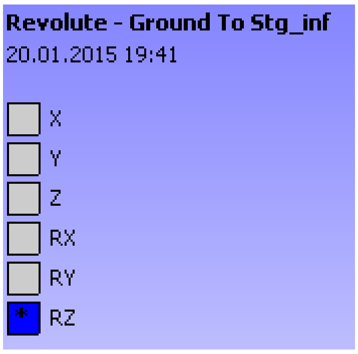
→ ↵ ? Static Structural (B5) → Inertial → Standard Earth Gravity (the selection of the gravitational acceleration implies the taking into account of the own weight of the metallic structure) → Details of "Standard Earth Gravity" → **Definition** → Direction: -Y Direction.

Insertion of the base connection joints

, Outline: ↵ Connections → Body-Ground → Revolute → Details of "Revolute - Ground To No Selection" → Mobile → Scope: [select Point 1 using the selection filter (Point)]. A rotating joint around the Oz axis is being considered

Details of "Revolute - Ground To No Selection"

Definition	
Connection Type	Body-Ground
Type	Revolute
Torsional Stiffness	0, N·mm/°
Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Coordinate System	Reference Coordinate System
Mobile	
Scoping Method	Geometry Selection
Scope	No Selection
Body	No Selection
Initial Position	Unchanged

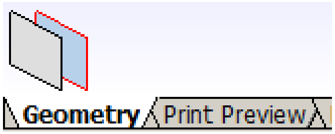


The operation will also be repeated for Item 8, in connection with the base.

Insertion of joints between segments

, Outline: ↵ Connections → Body-Body → Revolute → Details of "Revolute - No Selection To No Selection" → **Reference** → Scope: [Point 2 on the Dr_inf segment is selected using the selection filter (Point)] → Apply → **Mobile** → Scope: [the same Point 2 is selected, but which is located on the Dr_med segment, using the selection filter (Point)] → Apply.

When selecting the same point, the symbol below will appear in the lower left corner of the graphic window, which is the command button to toggle the selection of the two entities (segments, in this case) by clicking with the mouse on the two planes.



Details of "Revolute - No Selection To No Selection"	
Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Scoping Method	Geometry Selection
Scope	No Selection
Body	No Selection
Coordinate System	Reference Coordinate System
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Scope	No Selection
Body	No Selection
Initial Position	Unchanged
Behavior	Rigid

Details of "Revolute - Dr_inf To Dr_med"	
Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Scoping Method	Geometry Selection
Scope	1 Vertex
Body	Dr_inf
Coordinate System	Reference Coordinate System
Behavior	Rigid
Pinball Region	All
Mobile	
Scoping Method	Geometry Selection
Scope	1 Vertex
Body	Dr_med
Initial Position	Unchanged
Behavior	Rigid

Repeat the operation for the other torques in points 3, 4, 5, 6, 7 and obtain the torques shown in the tree below.

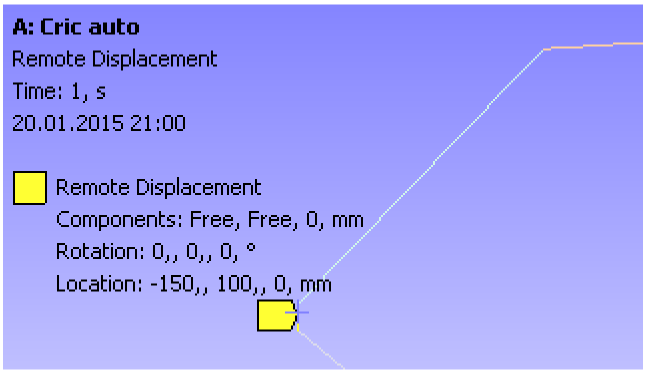
- Connections
  - Joints
    - Revolute - Ground To Dr_inf
    - Revolute - Ground To Stg_inf
    - Revolute - Dr_inf To Dr_med
    - Revolute - Dr_med To Dr_sup
    - Revolute - Dr_sup To Sup
    - Revolute - Sup To Stg_sup
    - Revolute - Stg_sup To Stg_med
    - Revolute - Stg_med To Stg_inf

Introduction of operating constraints

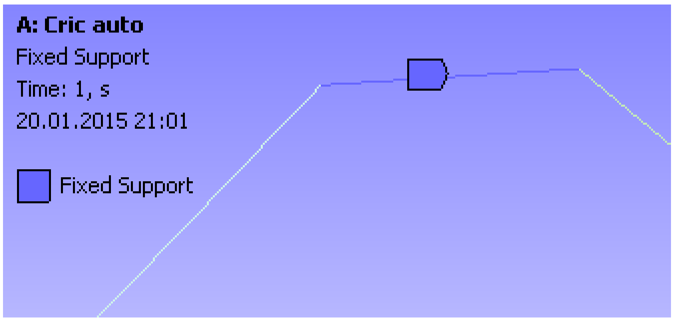
The jack will work taking into account the hypothesis that the median lateral segments will be able to move only in the xOy plane, without the possibility to rotate.

→ Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" → **Scope** → Geometry: [select the lateral segment *Dr_med*] → Apply; **Definition** → X Component: free, Y Component: free, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: 0.

The procedure will be repeated for the segment as well *Stg_med*.



→ ↲ ? Static Structural (B5) → Supports → Fixed Support → Details of "Fixed Support 2" → **Scope** → Geometry: [will be selected with ↲ the upper segment *Sup*] → Apply.

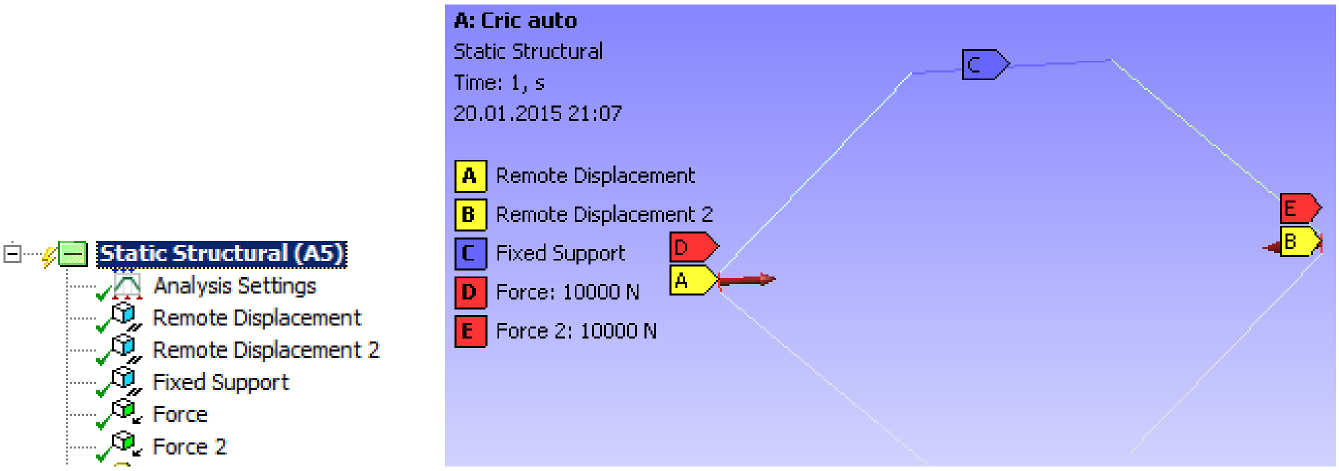


C.6 Load modeling

Input forces

→ ↲ ? Static Structural (B5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [will be selected with ↲ lateral segment *Dr_med*] → Apply; **Definition** → Define by: Components → X Component = 10.000 N, Y Component = 0, Z Component = 0. The procedure will be repeated for the *Stg_med* segment, changing only the direction of the force, towards the inside of the mechanism..

The constraints and loads of the structure will look like in the figure below



D. SOLVING THE FEA MODEL

D.1 Launching the calculation module and select the types of results

In order to select the final data types to be analyzed after the launch of the calculation module, the series of commands presented below will be followed.

→ ↳ Solution (B6) → Insert → Deformation → Total [use the commands in the command box open with ↳].

The same result can be obtained by using commands:

↲ Solution (A6) → Deformation → Total [the buttons in the menu bars are used] precum şi / and

↲ Solution (A6) → Deformation → Directional.

For this type of structure, the *Beam* tool can be applied in order to visualize the linearized stresses on the component elements. It is customary in the design of bar structures to take into account the components of axial stresses arising from the effect of axial and bending loads in all directions. The following are the other types of results to be analyzed:

↲ Solution (A6) → Tools → Beam Tool.

↲ Solution (A6) → Beam Results → Axial Force.

↲ Solution (A6) → Beam Results → Bending Moment.

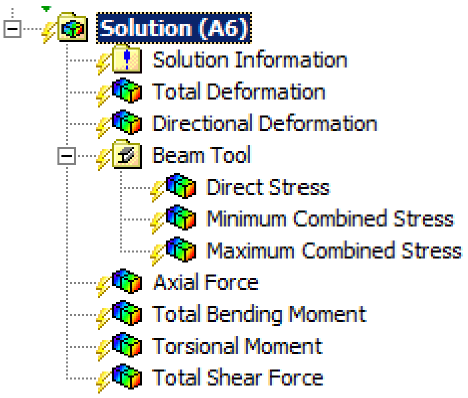
↲ Solution (A6) → Beam Results → Torsional Moment.

↲ Solution (A6) → Beam Results → Shear Force.

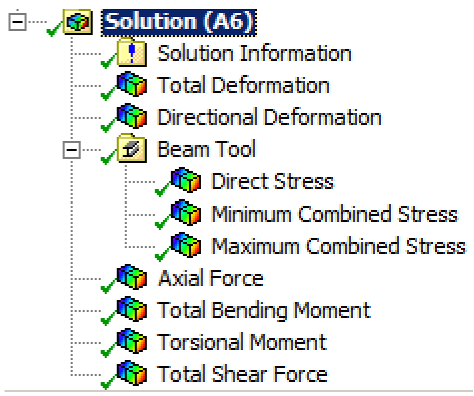
↲ Solution (A6) → Probe → Joint.

D.2. Launching the solving module

→ ↲ Solution (A6) → Solve.



→ Solve →



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu bars:

Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

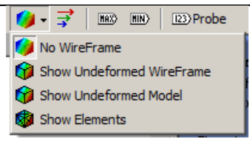
Total deformation view

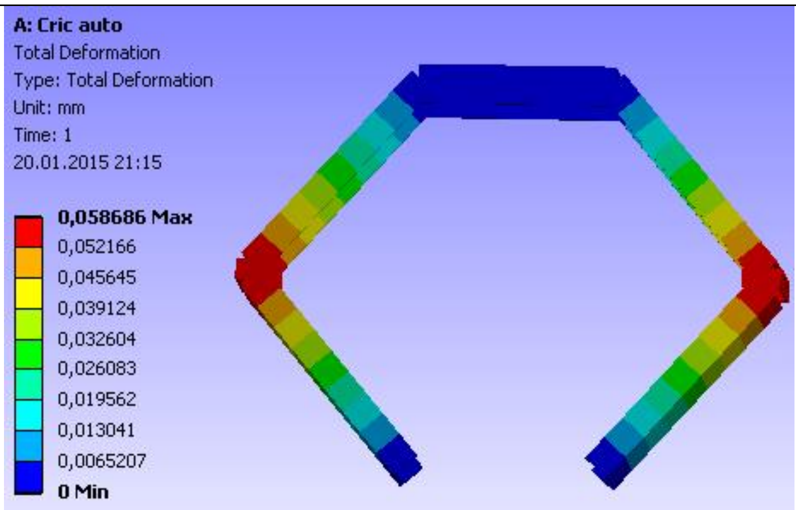
↲ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation.

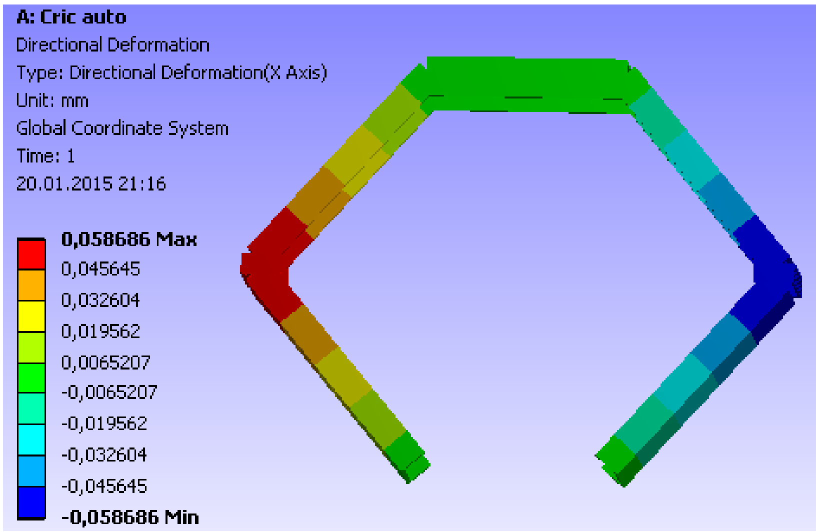
The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command .





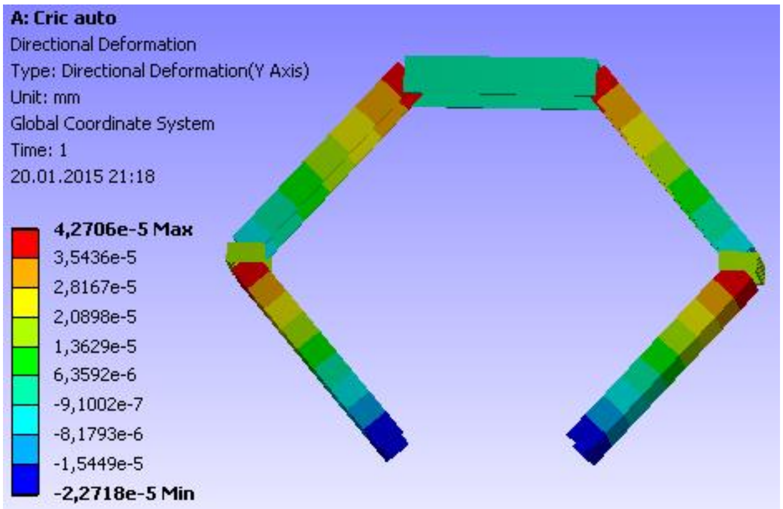
One-way deformation view

↲ Solution (A6) → Directional Deformation → Graph → Animation.



If you want to view it in another direction, follow the steps below:

↲ Solution (A6) → Directional Deformation → Details of "Directional Deformation" → **Definition** → Orientation : Y Axis → Solve.

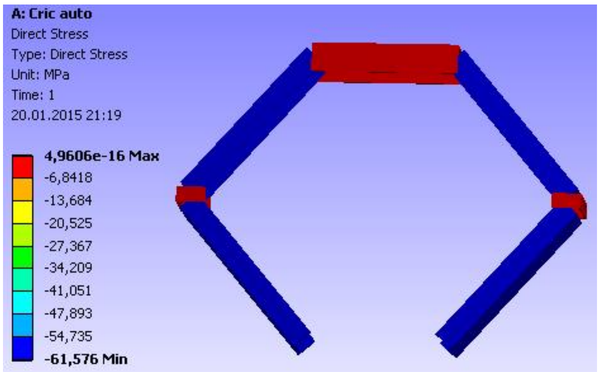


E.2. Visualize the fields of stress, forces and moments

Direct Stress

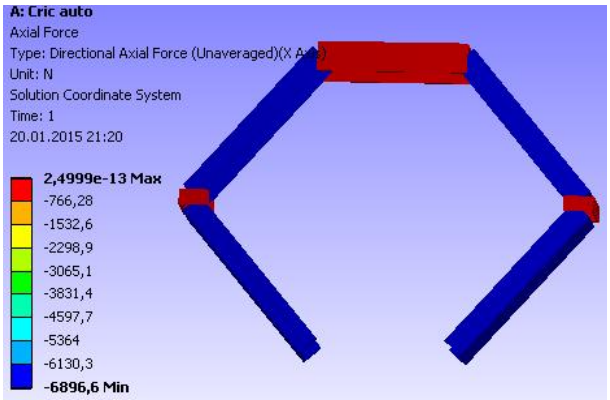
Direct Stress (σ_x) represents the component of the internal tension due to the axial force in an element of the mechanism.

Solution (A6) → Direct Stress → Graph → Animation.



Directional Axial Force

Solution (A6) → Axial Force → Graph → Animation.

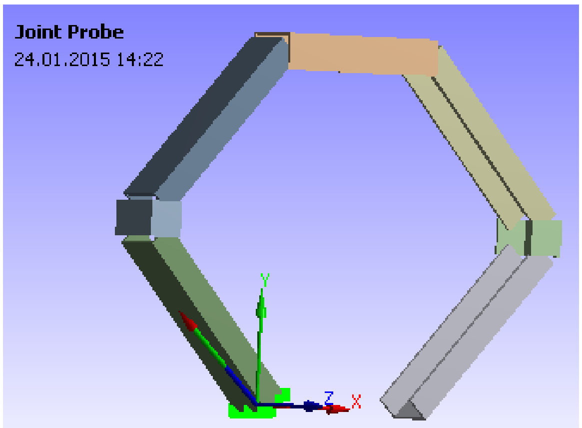


Joint Probe

Solution (A6) → Joint Probe → Tabular Data.

Tabular Data

	Time [s]	Joint Probe (Total Force X) [N]	Joint Probe (Total Force Y) [N]	Joint Probe (Total Force Z) [N]	Joint Probe (Total Force Total) [N]
1	1,	-5000,	4500,	1,6211e-005	6726,8



F. RESULTS ANALYSIS

It is observed that, despite the fact that the modeling of the articulated bar mechanism was performed with the help of one-dimensional bodies, the results obtained are suggestive, being presented in a 3D environment, due to the ease of the program used to attach various profiles to the structure. executed by the user. Modifying the profiles of the articulated bars is very easy to do, this can be done even at the end of an analysis, following that after an update command, the results of the new analysis will change according to the new initial conditions.
The realization of the rotational torques is easy, it is not necessary their 3D construction and the precise modeling of their geometry. The definition of the rotational torques can take into account the elastic characteristics of the joint. The positioning of the torques according to the bar profile can be chosen from several variants, offered by ANSYS.
From the point of view of the total deformations, it is observed that the maximum value is 0.05 mm in the area of application of the stress, in the direction of the Ox axis.
Examining the graphical representation of the axial stresses, it is observed that the lateral segments (2) are subjected to the compression stress - represented in blue. The information regarding the deformations, corroborated with the information regarding the internal stresses, the combined maximum stresses lead to the conclusion that the structure withstands loads without problems, the values of the maximum stresses not exceeding the allowed material limit (compression σ_{ac}=80 ... 100 Mpa).

Definition	
Connection Type	Body-Ground
Type	Revolute
Torsional Stiffness	0, N·mm/°
Torsional Damping	0, N·mm·s/°
Suppressed	No
Reference	
Coordinate System	Reference Coordinate System
Mobile	
Scoping Method	Geometry Selection
Scope	1 Vertex
Body	Inf_stg
Initial Position	Unchanged
Behavior	Rigid
Pinball Region	Rigid Deformable Beam (Beta)
Stops	

G. CONCLUSIONS

From the point of view of the pre-processing phase, it can be seen that the use of 1D bodies involves minimal resources for both modeling and discretization. Another strong point is that the transverse profile of the sections can be modified / oriented very easily, without influencing the basic shape of the bar structure. Moreover, it is possible to use different profiles for each section. The sections can be connected in several ways, depending on the central axis of the profiles used.
The introduction of supports, constraints and demands is quick and easy. The declaration of the materials, as well as the discretization of the bar structure are controllable processes, which can be done automatically or manually.
Analyzing the results obtained by MEF, it can be seen that it provides much more data, at a time and with much lower resource consumption, than the analytical version. It can be seen that the structure of the beams is very little required, and smaller profiles can be used in order to achieve savings. Changing the profile of the beam sections and recalculating is done in a very short time, being an easy procedure.

Application: FEA-A.17
Dynamic analysis of collision

KEY WORDS
Dynamic Analysis, Plane Geometric Model, Plane Stress State, Linear Material, 1D Finite Element, Linear Finite Element, Machine Element, Collision

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

The Ansys Workbench E*xplicit dynamics* suite it enables to capture the physics of shortduration events for products that undergo highly nonlinear, transient dynamic forces. In many cases, the accuracy of an explicit solution can be verified only via comparison with physical experiments. For some problems (such as explosions), it may be too expensive or impossible to perform tests.

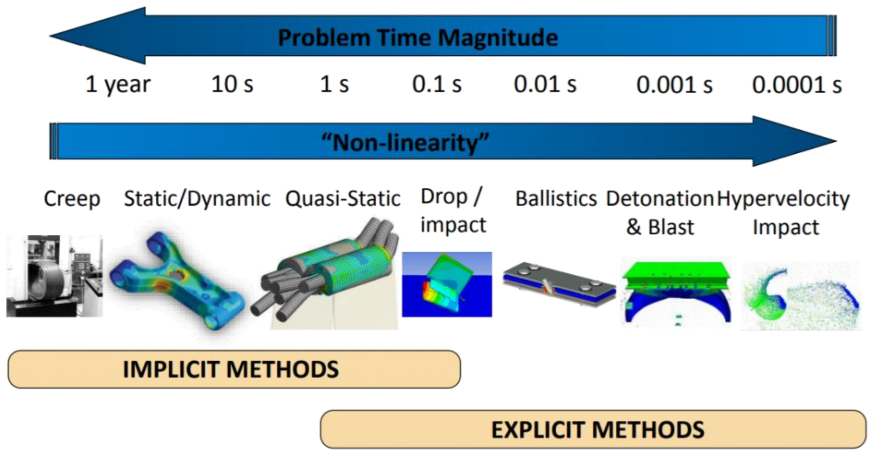
“Implicit” and “Explicit” refer to two types of time integration methods used to perform dynamic simulations. Explicit time integration is more accurate and efficient for simulations involving – Shock wave propagation – Large deformations and strains – Non-linear material behaviour – Complex contact – Fragmentation – Non-linear buckling. Typical applications are drop tests, impact and penetration. ANSYS Explicit Dynamics analysis software provides simulation technology to help simulate structural performance long before manufacture.

A time integration method used in Explicit Dynamics analysis system. It is so named because the method calculates the response at the current time using explicit information. After defining the initial conditions (initial velocity, angular velocity), the analysis setting has to be maintained as per the problem requirement. In the analysis setting, time steps have to be defined explicitly, including:

- Initial time step
- Minimum time step
- Maximum time step
- Time step safety factor

In case of drop test the standard earth gravity is also taken into account.

ANSYS Explicit Dynamics utilizes the Autodyn solver within the standard ANSYS Mechanical interface to analyses transient structural events and it is used for simulating fracture, cutting, failure, buckling, impact, drop as well as highly nonlinear quasi-static simulations that the implicit APDL-based solvers would struggle to converge.



https://www.mechead.com/what-is-explicit-dynamics-in-ansys/

A.2 Application description

In practice, there are many mechanical phenomena that are manifested by mechanical contacts and stresses made in very short periods of time, in the form of collisions. Some of these collisions can cause elastic deformation of the parts in contact, others can cause plastic deformation or even destruction and expulsion of material (in the case of penetration phenomena). In the field of motor vehicles, these dynamic impact requirements are very common. An analysis of the impact phenomena between a body element and a static element is very suggestive, and can be used in the design stages of body elements and passive safety elements.

A.3 Application goal

The application aims to determine the maximum values of the fields of displacements, deformations, internal stresses produced in collision on the component elements.

For this analysis, the use of two-dimensional elements was considered due to the simplicity of the geometric construction and the ease of modifying the profile of the studied elements.

B. PREPARATION OF THE MODEL FOR ANALYSIS

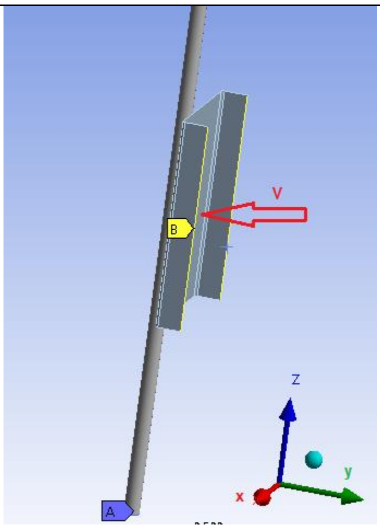
B.1 The model definition

In order to draw up the finite element analysis model associated with the present application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight, speeds, accelerations),
- material characteristics.

B.2 The analysis model description

The analyzed model is intended to be very simple, consisting of only two elements: one of the tubular type which represents a fixed obstacle (steel pipe pillar with an average diameter of 60 mm and the wall of 5 mm) and the other, of the panel type by alluminium alloy 1 mm thick sheet, representing the moving object (dimensions are presented in Chapter C.3 - Geometric modeling). The collision with the vertical pole is considered to take place in the normal direction of the sheet metal panel, in the middle of it, according to the drawing below. The metal pillar is embedded at the bottom and the sheet metal panel moves in the direction of Ox in the direction of approaching the pillar with a speed of 25 m / s.



- point A – TX TY TZ RX RY RZ
- Vertical edges of the panel B – TX RY
- v_y = -25 m/s

B.3 Characteristics of the material

For finite element analysis the strength characteristics of the material, S235 steel are:

- modulus of longitudinal elasticity, E = 204,000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.3.

The characteristics of the second material, aluminum alloy, remain unchanged, according to the software library of materials.

- modulus of longitudinal elasticity, E = 75.000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.32.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Create of the project

WB, Toolbox: ↵ ⊟ Analysis Systems → ↵↵ Explicit Dynamics (the subproject window appears automatically); → [the name can be changed Explicit Dynamics în *Collision*].

Problem type setting (3D)

A : ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from list ↵▼, ↵3D] → [close window ↵✕].

Saving the project

↵ Save As... → Save As, File name: [input name, *Collision*] → ↵ Save.

C.2 Modelling of material characteristics

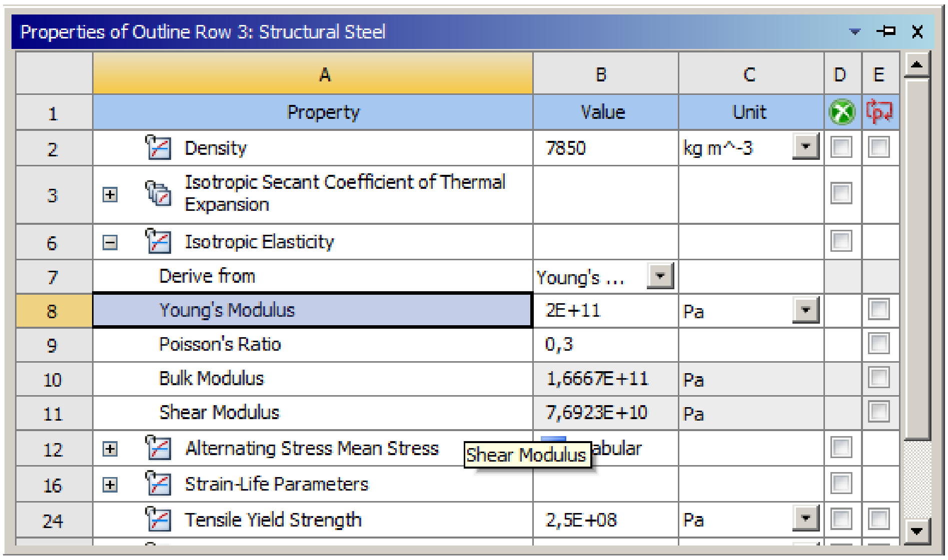
As there will be two separate parts, two different materials will be considered: steel and aluminum.

, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, Young's Modulus, [select from column C (Unit) cu ↵▼, ↵MPa], [input in column B (Unit) value, 204000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

To introduce the second material, follow the steps:

Engineering Data ✓ → Engineering Data Sources → Engineering Data Sources → General Materials ☐ → ☑ (se bifeaza căsuța / check box) → Outline of ANSYS GRANTA Materials Data for Simulation (Sample) → Aluminum alloy, wrought, 6061, T6 (select Al alloy by check the box) → Outline of Schematic A2: Engineering Data (both materials are active: steel and aluminum).

If the window Properties of Outline Row 3: Structural Steel is not visible, the Outline and Properties or Reset Workspace options will be activated in the View menu, then the *Engineering Data Sources* command .


Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	Alternating Stress Mean Stress	Shear Modulus abular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

C.3 Geometric modelling

C.3.1 Loading DesignModeler (DM) module

, Project Schematic: ↳ Geometry → ↵ New DesignModeler Geometry... → Units: ↵ ✓ Millimeter, ↵OK.

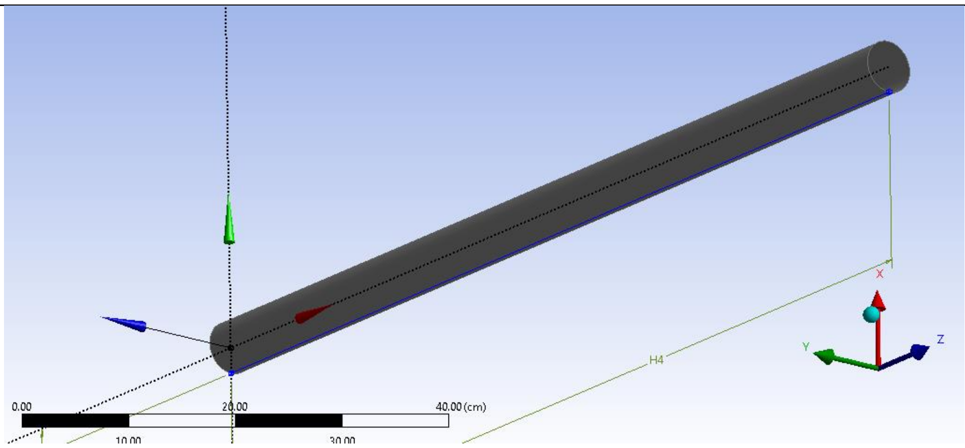
C.3.2 Pillar modelling

Construct a pipe with a circular section along the OZ axis, with surface-type elements, as follows:

Tree Outline → ZXPlane → Sketching (Look at plane) (draw a line parallel to the OZ axis, at distance 25 mm, 1000 mm long).

Dimensioning commands in the menu Dimensions will be used for sizing and positioning.

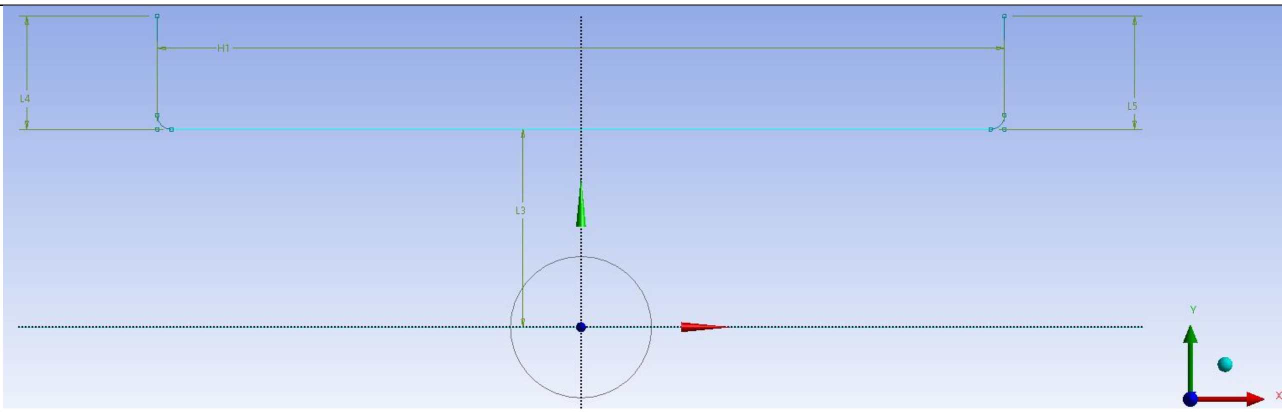
To create the 3D model, use the command Revolve, then rotate the drawn segment around the OZ axis, then Generate.



C.3.3 Panel modelling

A rectangular panel is constructed in a plane parallel to the XOZ, using surface-type elements. Since the panel will have reinforcements on two edges, its profile will be drawn in a plane perpendicular to the OZ and XOY axis, respectively.

Tree Outline → XYPlane → Sketching (Look at plane) Polyline (draw the profile in the figure below).



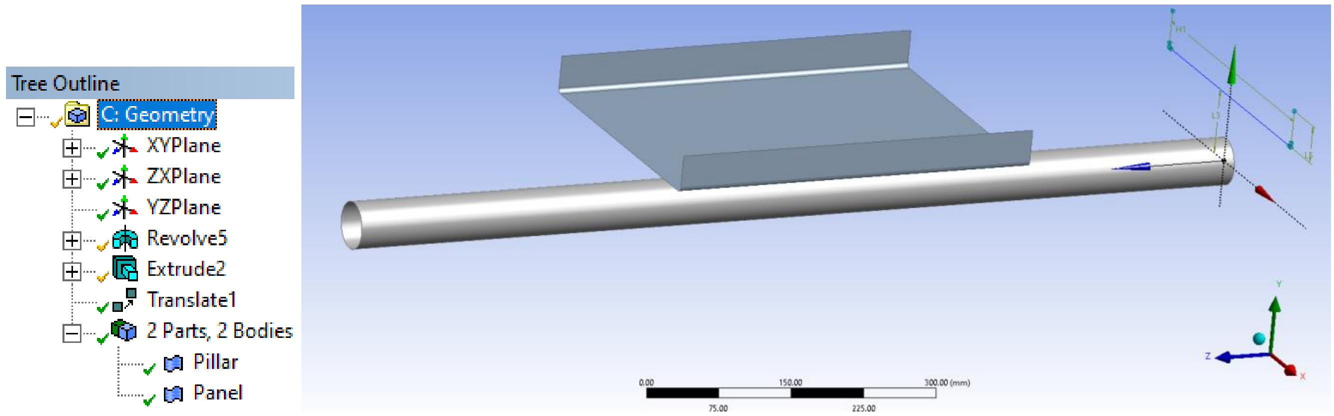
H1	300 mm
L3	70 mm
L4	40 mm
L5	40 mm

Modify

Fillet Radius: 5 mm

The profile will be extruded with the Extrude command in the OZ direction on the length of 400 mm, then it will be translated in the OZ direction at half the height of the pole, respectively 300 mm, using the commands:

Create → Body Transformation → Translate → Generate.



C.3.4 Saving the geometric model

DM → (Save Project) → <u>File</u> → Close Design Modeler.

C.4 Finite element modelling

C.4.1 Activate the discretization module and set the finite element type

<u>Launching of the finite element modeling module</u>

WB, Project Schematic ↵↵ Model → launching module *Mechanical [ANSYS Multiphysics]*.

<u>Setting the unit of measurement system</u>

M: ↵ Units → ↵ Metric (mm, kg, N, s, mV, mA) (the system of units of measurement is usually set by default).

<u>Setting the material characteristics</u>

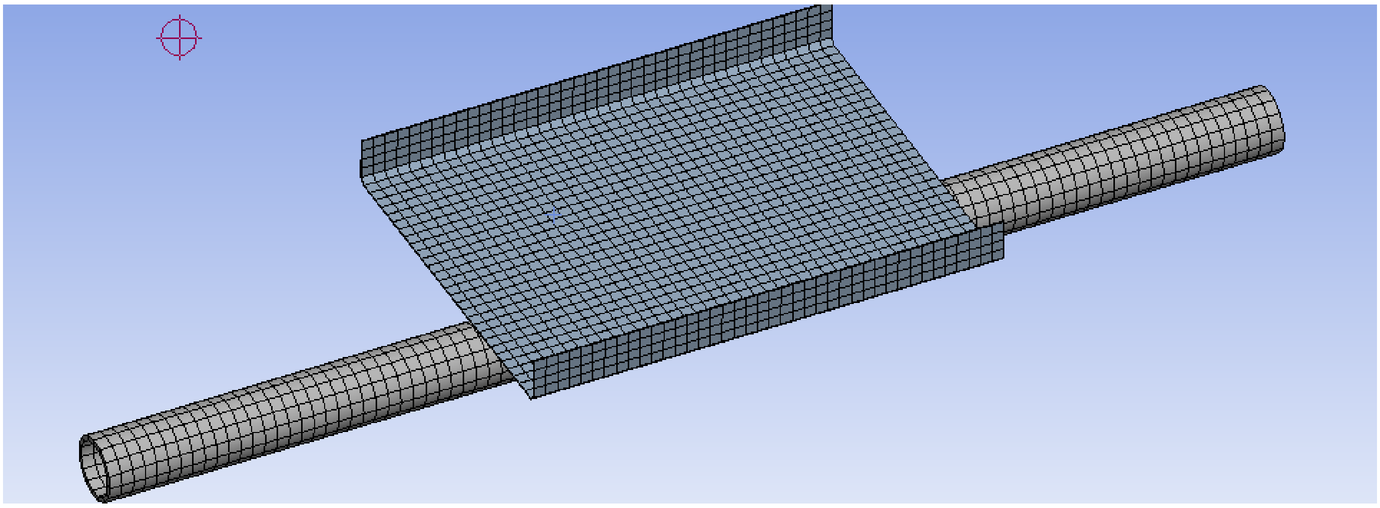
Outline: ↵ Geometry → ? Pillar → Details of "Pillar" → **Material** : ↵Assignment → [select from list ↵, ↵ Structural Steel (default)] → **Definition** → Thickness 4.e-003 m .

Outline: ↵ Geometry → ? Panel → Details of "Panel" → **Material** : ↵Assignment → [select from list ↵, ↵ Aluminum Alloy (default)] → **Definition** → Thickness 1.e-003 m .

C.4.2 Model discretization and finite element size setting

M, Outline: ↵ Mesh → **Sizing** → Details of "Edge Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ 3D geometry of the pillar, using the selection filter (Body)] Apply; **Definition Element** → Size: 10 mm → **Update** .

The operation will be repeated for the other body as well – Panel, Size: 10 mm.

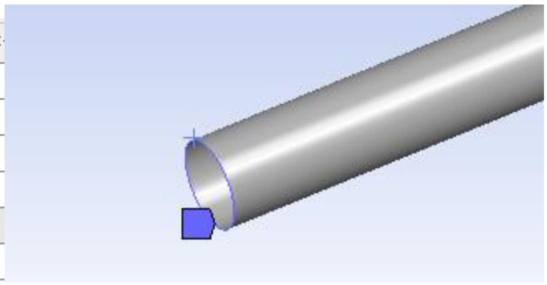


C.5 Modeling joints and constraints

Inserting the fixed support

Outline: Explicit Dynamics ↵ Fixed → Details of "Fixed Support" → Geometry → Scope: [the circle at level 0 corresponding to the pillar is selected using the selection filter (Edge)].

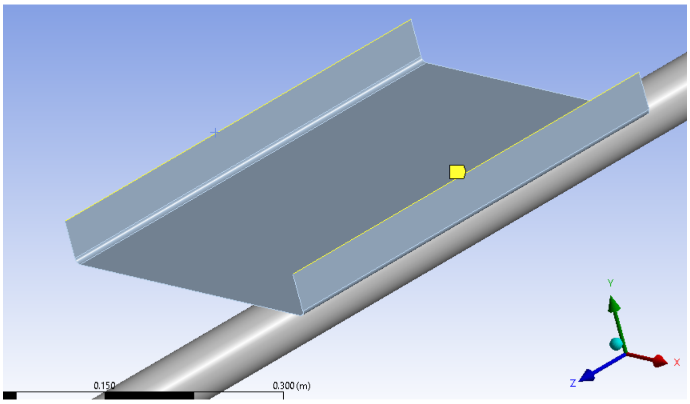
Details of "Fixed Support"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Edge
Definition	
Type	Fixed Support
Suppressed	No



Enter the movement conditions of the panel

Outline: ↵ Explicit Dynamics → Displacement Structural → Geometry → Scope: [select the edges parallel to the OZ axis of the panel using the selection filter (Edge)] → OX, OZ = 0, OY = free.

Details of "Displacement"	
Scope	
Scoping Method	Geometry Selection
Geometry	2 Edges
Definition	
Type	Displacement
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. m (step applied)
Y Component	Free
Z Component	0. m (step applied)
Suppressed	No

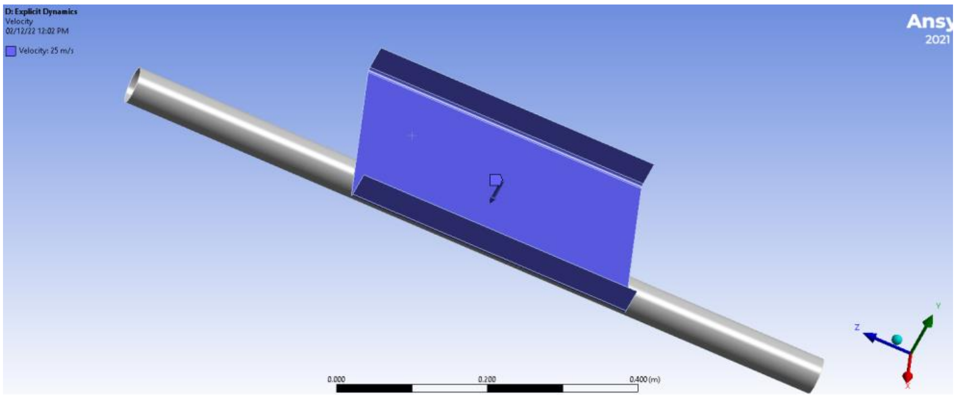


C.6 Load modeling

Entering the speed of the panel

Outline: Explicit Dynamics ↵ Initial Conditions → Velocity T=0 → Details of "Velocity" → **Scope** → Select Geometry: [will be selected with ↵ 3D geometry of the panel, using the selection filter (Body)] Apply; → **Definition** → Define by: Components → x = 0, y = -25 m/s, z = 0.

Explicit Dynamics (D5)
- Initial Conditions
  - Pre-Stress (None)
  - Velocity

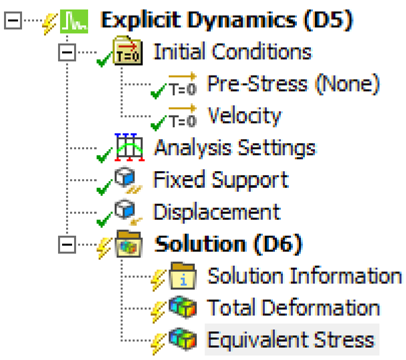
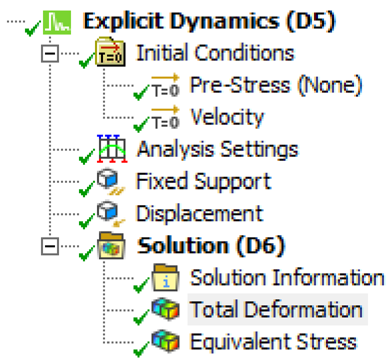


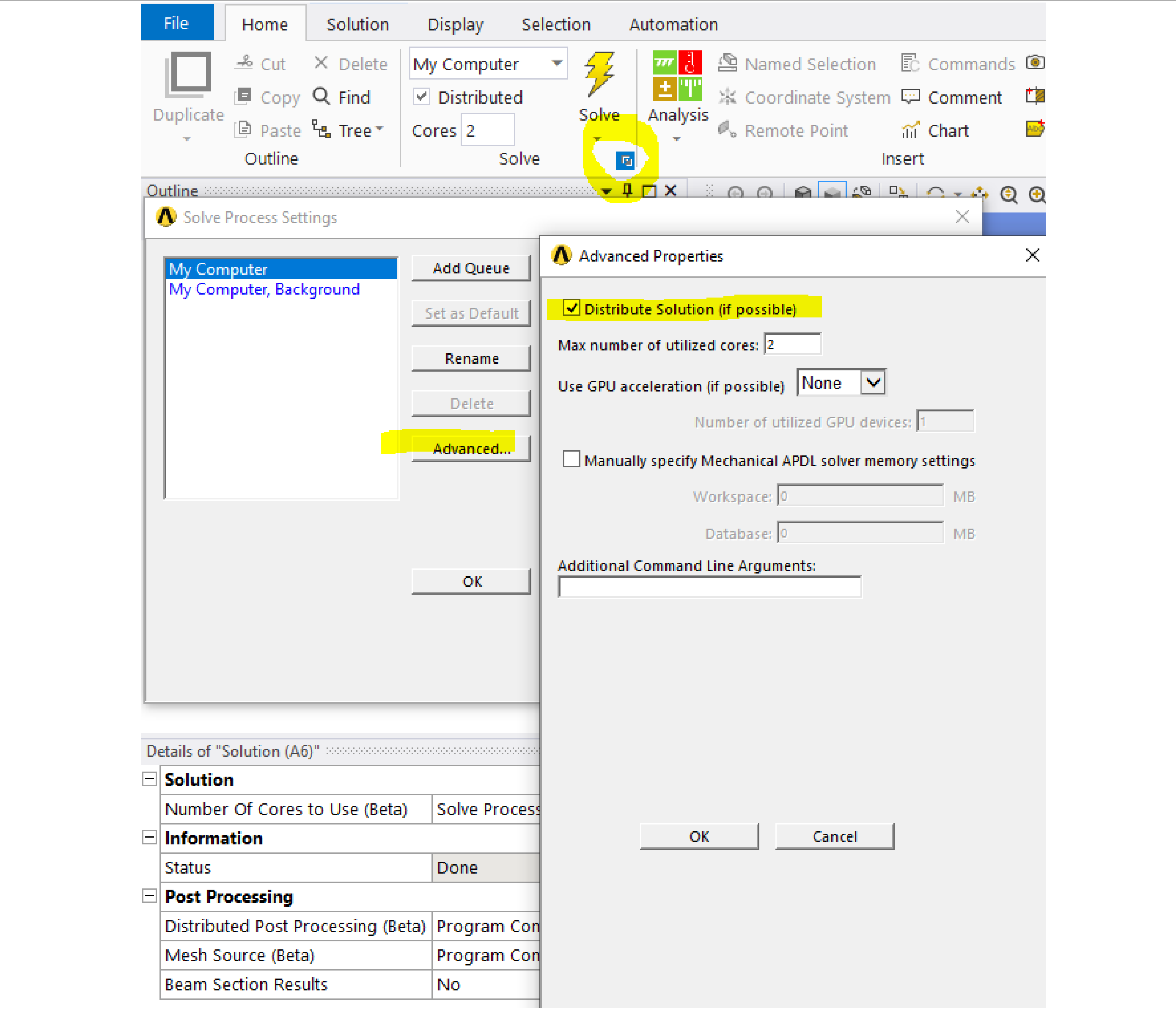
Introduction of analysis time

, Outline : Explicit Dynamics ↵ Analysis Settings → Details of "Analysis Settings" → **Step Controls** →

End Time: 0.01 s (estimate the travel time of the panel to the collision with the pole, depending on its initial distance and speed: $t = dist/speed = 70\ mm / 25\ m/s = 0.0028\ s$).

D. SOLVING THE FEA MODEL

D.1 Launching the calculation module and select the types of results
In order to select the final data types to be analyzed after the launch of the calculation module, the series of commands presented below will be followed.
→ ↳ **Solution (D6)** → Insert → Deformation → Total [use the commands in the command box open with ↳].
The same result can be obtained by using commands: ↵ **Solution (D6)** → **Solution** → Deformation → Total Deformation [the buttons in the menu bars are used] and ↵ **Solution (D6)** → **Solution** → Stress → Equivalent (von-Mises). ↵ **Solution (A6)** → Force Reaction → Tabular Data.
D.2. Launching the solving module
→ ↵ **Solution (D6)** → Solve.  → Solve → 
In case of errors in the processing of the model of the type "*An unknown error occurred during solution ...*", the solution below must be tried, by unchecking the highlighted button.



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu bars:

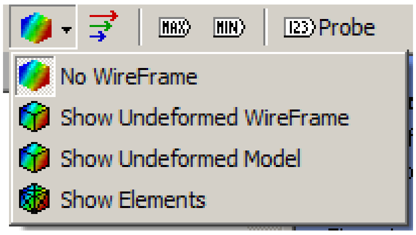
Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

Total deformation view

↵ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

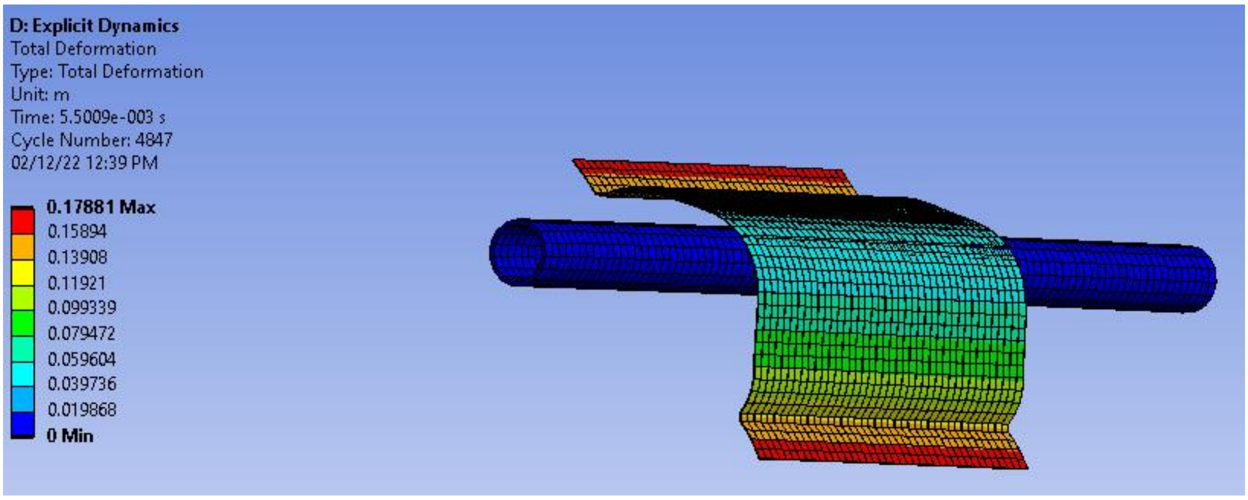
Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation.



The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command .

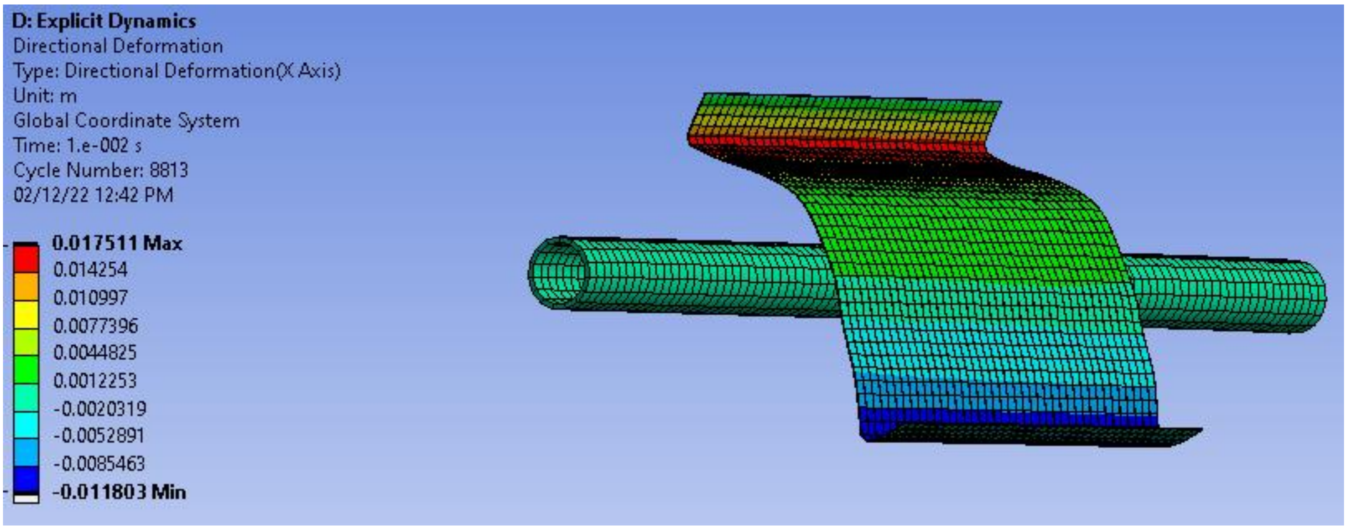
Total deformation view

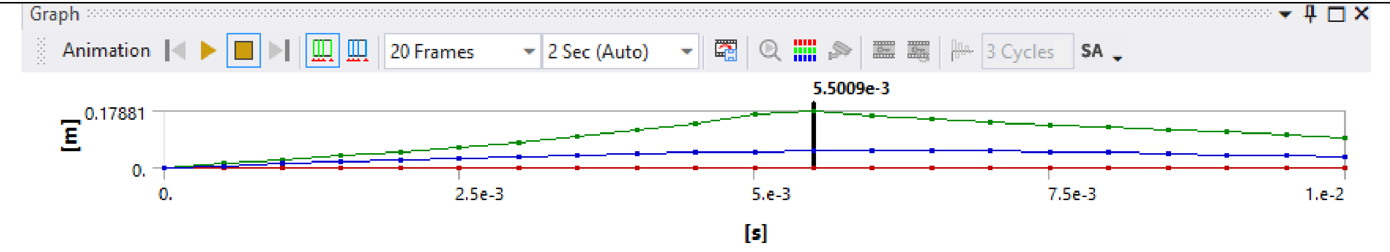
↵ Solution (A6) → Total Deformation → Graph → Animation.



If you want to view it in another direction, follow the steps below:

↵ Solution (A6) → Directional Deformation → Details of "Directional Deformation" → **Definition** → Orientation : Y Axis → Solve.

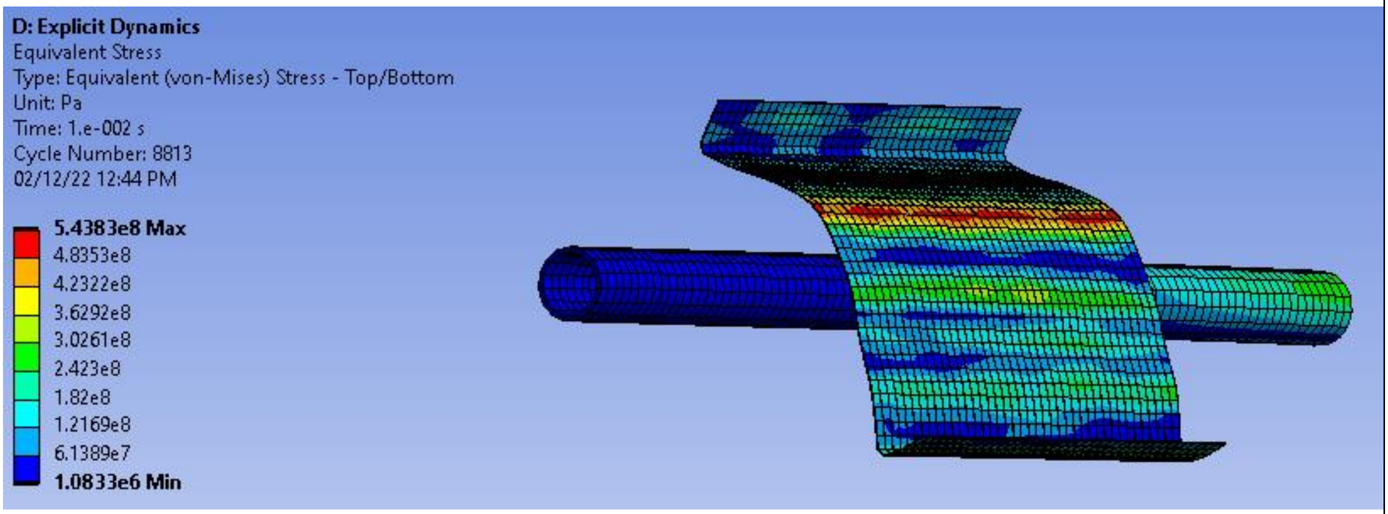


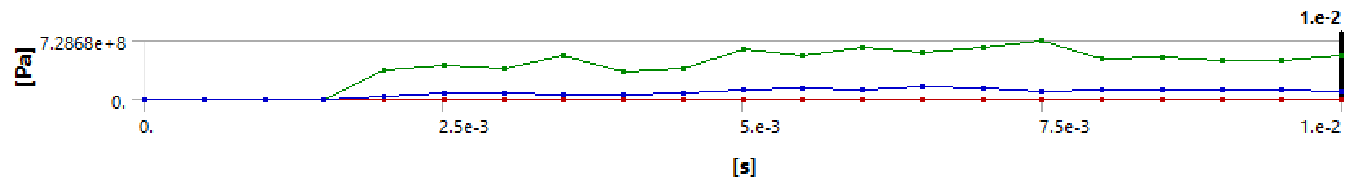


E.2. Visualizing the fields of stress, forces and moments

Equivalent Stress

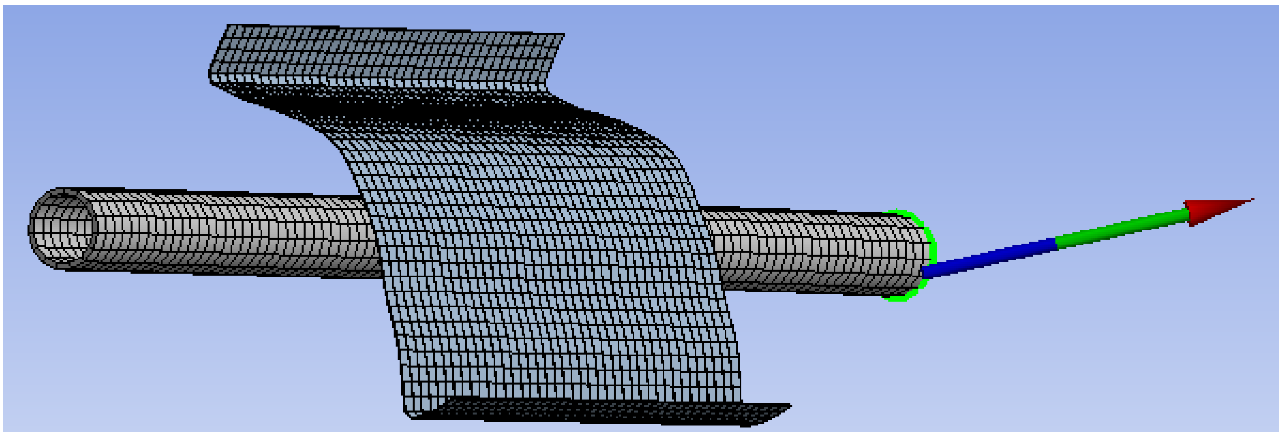
Solution (A6) → Equivalent Stress → Graph → Animation.





Force reaction

Solution (A6) → Force Reaction → Tabular Data.



	Time [s]	Force Reaction (X) [N]	Force Reaction (Y) [N]	Force Reaction (Z) [N]	Force Reaction (Total)
13	6.0003e-003	-328.63	2307.3	659.87	2422.2
14	6.5007e-003	464.37	5059.5	-4897.5	7056.9
15	7.001e-003	335.72	6727.6	-1172.2	6837.2
16	7.5002e-003	-461.69	2355.4	-606.84	2475.8
17	8.0005e-003	417.92	483.06	2291.	2378.3
18	8.5008e-003	-43.553	1236.6	2040.5	2386.4
19	9.0009e-003	-803.21	-326.26	-1053.	1364.
20	9.5011e-003	139.2	2150.4	1931.9	2894.1

F. RESULTS ANALYSIS

It is observed that, despite the fact that the modeling of the parts was performed with the help of surface type bodies, the results obtained are suggestive, being presented in a 3D environment, due to the ease of the program used to attach various thicknesses to the structure.

Changing the thickness and materials of the various components is very easy to do, this can be done even at the end of an analysis, and after an update order, the results of the new analysis will change according to the new initial conditions.

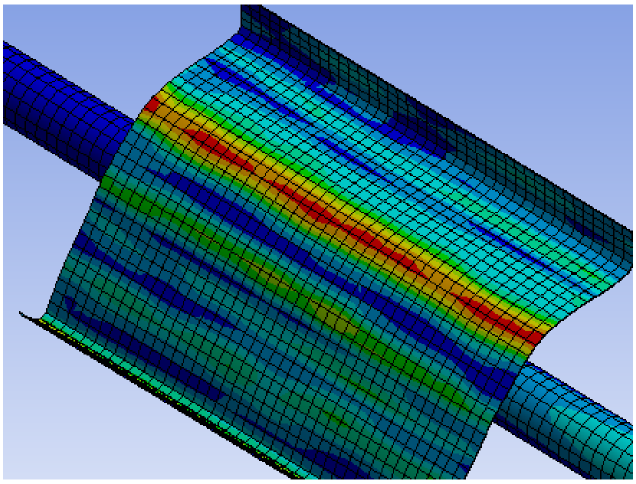
From the point of view of the total deformations, it is observed that the maximum value is 0.0175 m in the area of the vertical edges, the entities furthest from the center of the panel.

Examining the graphical representation of the total equivalent stresses (fig.a), it is observed that the panel is strongly stressed in the areas of contact with the column, reaching values (543 MPa) that exceed the elastic limit of the material, entering the flow area (□c=280 Mpa).

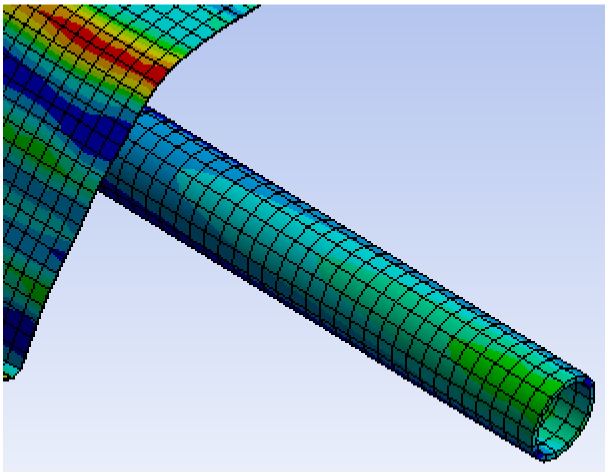
In other words, at these initial data (constraints and speed of the panel) a plastic deformation of the panel can be noticed.

From the point of view of the tubular pillar, made of steel (fig. b), a voltage of values of 180-200 Mpa is observed, below the flow limit. So, the pillar will deform elastically, returning to its original shape and dimensions.

If the initial speed changes from 25 to 10 m / s, it will be noticed that the value of the stresses generated by the impact is lower (280 Mpa), as well as the deformation of the panel (0.005 m), remaining in the area of elastic deformations.



a.



b.

G. CONCLUSIONS

From the point of view of the pre-processing phase, it can be seen that the use of 2D bodies involves minimal resources for both modeling and discretization. Another strong point is that the thickness of the parts (either of the column or of the sheet metal panel) can be modified very easily, without influencing the basic shape of the bar structure.
The introduction of supports, constraints and demands is quick and easy. The declaration of materials, as well as the discretization of the structure in the form of surfaces are controllable processes, and can be done automatically or manually.
Analyzing the results obtained by FEM, it can be seen that it provides much more data, at a time and with much lower resource consumption, than the analytical version.

References

(1). Akin, J.Ed, Finite Element Analysis Concepts via SolidWorks, Lecture Notes, Rice University, Houston, Texas, 2009.

(2). Al Makky, A.: Tutorial 3: Working with DesignModeler, http://www2.warwick.ac.uk/fac/sci/eng/pg/students/esrhaw

(3). Allen, G.: Geometric Modeling Problems in Industrial CAD/CAM/CAE, Siemens PLM Software, Shanghai.

(4). ALTAIR HyperWorks 12.0, User Guide, http://www.altairhyperworks.com.

(5). Ananthasuresh, G.K., The Principle of Minimum Potential Energy, lecture notes, chapter 2, Department of Mechanical Engineering, Indian Institute of Science, Bangalore, http://www.mecheng.iisc.ernet.in

(6). ANSYS Workbench 14.0 User Guide // DesignModeler User Guide // Introduction / MechIntro /

(7). Barton, M., Rajan, S.D.: Finite Element Primer for Engineers, lecture notes, Arizona State University, 2002.

(8). Bathe, K.J.: Finite Element Analysis of Solids and Fluids II » Lecture Notes, http://ocw.mit.edu/courses/mechanical-engineering/2-094-finite-element-analysis-of-solids-and-fluids-ii-spring-2011/lecture-notes/

(9). Bonet, J., Wood, R.D.: Nonlinear continuum mechanics for Finite Element Analysis, Cambridge University Press, ISBN 0 521 57272 X hardback, 1997.

(10). Bremar Automotion Pty Ltd: Finite Element Analysis Revealed: uncovering engineering's latest design tools element, www.bremarauto.com.

(11). Cailletaud, G., El Arem , S.: Introduction to Finite Element Method, Centre des Materiaux, MINES Paris Tech, UMR CNRS 7633, WEMESURF course, Paris, 2009.

(12). Catbas, N.: Finite Element Analysis (Overview), lecture notes, CES 6116, University of Central Florida.

(13). Chatzi, E.: The Finite Element Method for the Analysis of Non-Linear and Dynamic Systems, lecture notes, Swiss Federal Institute of Technology Zurich, 2010.

(14). Dassault Systemes, Tutoriale CATIA, http://www.3ds.com/support/ documentation/users-guide/

(15). Dixit, U.S., Finite Element Method: an introduction, Department of Mechanical Engineering, Indian Institute of Technology Guwahati, lecture notes, 2007.

(16). Faur, N.: Elemente finite. Fundamente, Editura POLITEHNICA, Timişoara, ISBN 973-8247-98-5, 2002.

(17). Felippa, C.A.: Introduction to Finite Element Methods, course, University of Colorado at Boulder, (downloaded from http://www.colorado.edu/engineering/CAS/courses.d/IFEM.d/Home.html , oct. 2013).

(18). http://cadcamfunda.com/cadcam_softwares

(19). http://usa.autodesk.com/adsk/servlet/item?siteID=123112&id= 17670721

(20). http://www.simscale.de/_en/index.php?v=2&page=index

(21). http://www.sv.vt.edu/classes/MSE2094_NoteBook/97ClassProj/num/widas/history.html

(22). http://www.z88.de/

(23). Hall, C.: Finite Element Course – A training Manual with Worked Examples used in Industry, www.value-design-consulting.co.uk .

(24). Hutton, D.V.: Fundamentals of Finite Elelement Analysis, McGraw-Hill, ISBN 0-07-239536-2, 2004.

(25). Knight, C.E., The Finite Element Method in Mechanical Design, PWS-KENT, 1993.

(26). Kreith, F.: The CRC handbook of mechanical engineering, CRC Press, p. 15-1, ISBN 978-0-8493-9418-8, 1998.

(27). Liu, G.R., Quek, S.S.: The Finite Element Method. A practical course, Butterworth-Heinemann, 2003.

(28). Liu, Y.: Introduction to the Finite Element Method, lecture notes, University of Cincinnati, 2003.
(29). Manopulo, N.: An Introduction to the Finite Element Analysis, lecture notes, Joint Advanced Student School, St. Petersburg, 2005.
(30). Marin, C., Hadâr, A., Popa, I.F., Albu, L.: Modelarea cu elemente finite a structurilor mecanice, Editura Academiei Române şi Editura AGIR, Bucureşti, 2002.
(31). Micu, S.D.: Introducere în metoda elementului finit, Universitatea din Craiova, grant CNCSIS 80 / 2005.
(32). Moaveni, S., Finite Element Analysis – Theory and Application with ANSYS, 2nd Ed., Pearson Education, 2003.
(33). Mogan, Gh., Butnariu, S., Analiza cu elemente finite. Aplicatii in CATIA, Ed. Universităţii Transilvania, 2007, ISBN 978-973-598-159-4
(34). Narayan, K. Lalit: Computer Aided Design and Manufacturing. New Delhi: Prentice Hall of India. p. 3. ISBN 812033342X, 2008.
(35). Nestorovic, G.: Principles of computer modelling of the solid products learning, Interdisciplinary Description of Complex Systems 6(1), 67-73, 2008.
(36). Nikishkov, G.P.: Introduction to the Finite Element Method, lecture notes, UCLA, 2001.
(37). Pascu, A.: Metoda elementului finit, curs, Catedra Organe de maşini şi Tribologie, Universitatea Politehnica Bucureşti, 2002.
(38). Qi, H.: Finite Element Analysis, lecture notes, MCEN 4173/5173, 2006.
(39). Rao, S.S.: The Finite Element Method in Engineering, Elsevier Science & Technology Books, ISBN: 0750678283, 2004.
(40). Roensch, S.: The Finite Element Method:A Four-Article Series, available at http://www.finiteelement.com, dec. 2013.
(41). Sadd, M.H.: Introduction to Finite Element Methods, lecture notes, MCE 565, Wave Motion & Vibration in Continuous Media Spring, 2005.
(42). Segerlind, L.J.: Applied Finite Element Analysis, second edition, John Willey and Sons, 1984.
(43). Siemens: White Paper - Buyer’s guide for pre- and postprocessing software, , Issued by: Siemens PLM Software. © 2012. Siemens Product Lifecycle Management Software Inc.
(44). Sorohan, Şt.: Elemente finite în ingineria mecanică, curs, Universitatea Politehnica Bucureşti, 2006.
(45). Suvranu De, Introduction to Finite Elements – MANE 4240/CIVL 4240, lecture notes, 2013.
(46). Zielinski, T.G.: Introduction to the Finite Element Method, Introductory Course on Multiphysics Modelling, ICMM lecture, 2007.
(47). Zienkiewicz, O.C. and Taylor, R.L., The Finite Element Method, Fifth Edition, vol. 1, 2, 3, Butterworth – Heinemann, 2000.
(48). de Weck, O., Finite Element Method, Lecture notes, Engineering Design and Rapid Prototyping, Massachusetts Institute of Technology, 2004.
(49). Yücel, H., Introduction to discontinuous Galerkin finite element methods (DG-FEMs), Computational Methods in Systems and Control Theory Max Planck Institute for Dynamics of Complex Technical Systems Magdeburg, Dec 12th, 2012.