Application: AEF-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
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 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 .

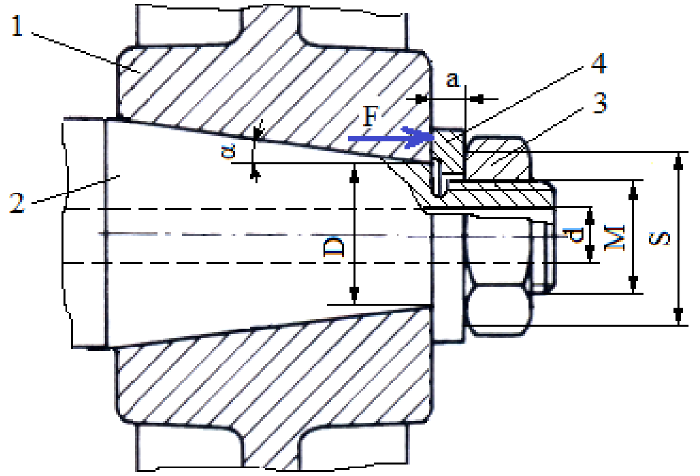
A.2 Application description

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.

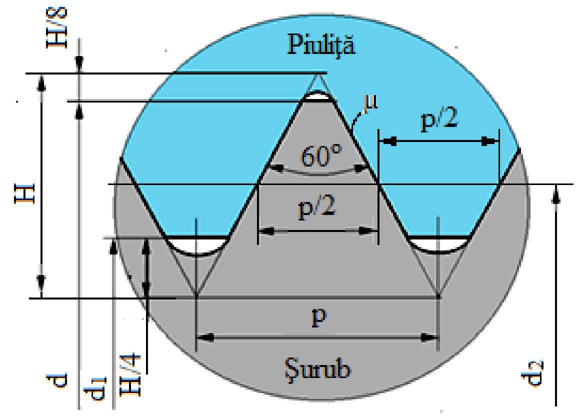
The metric fixing threads have the profile angle 60° and the theoretical height H = 0.866 p, where p is the thread pitch. The contact surfaces are delimited by cylindrical surfaces with diameter d1 on the inside and diameter d_2 on the outside, respectively.

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, $\alpha = 10^0$. 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

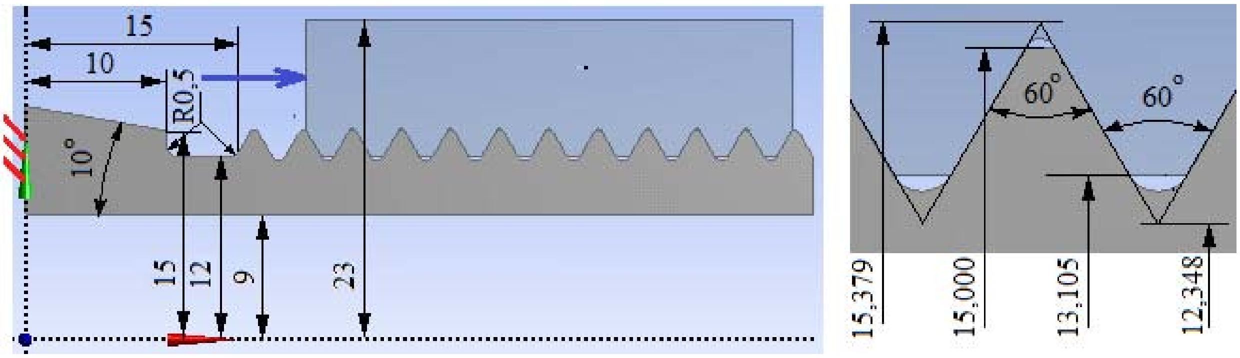
B.1 The model definition

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

Creating of the project

Λ, Toolbox: ↵⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural in *AEF-A.8*].

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 ↵ ✕].

Save of the project

↵ Save As… → Λ Save As, File name: [input name, AEF-A.8] → ↵ Save.

C.2 Modelling of material and environment characteristics

Λ, 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)

Λ, Project Schematic: ↳ Geometry → ↵ DM **New Geometry…** → ANSYS Workbench: ↵ ◉ Millimeter, ↵OK.

C3.2 Sketch generation, screw

Viewing default plane (XY)

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

Generating of horizontal and vertical lines

↵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).

Cuting lines at the edge

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

<u>*Partial sketch dimensioning*</u>

Dimensioning in the horizontal direction

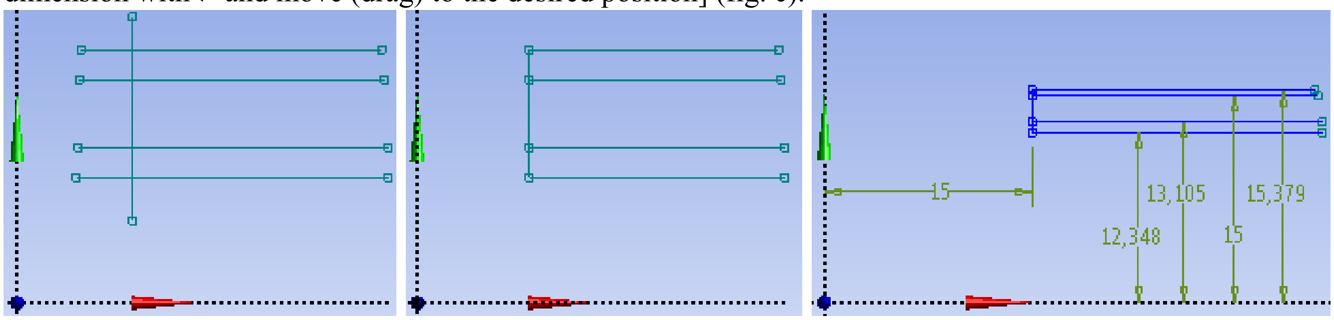
Sketching Toolboxes: ↲ Dimensions → ↲ Horizontal → [select with ↲ the line parallel to the Y axis and the Y axis] (the dimension is automatically displayed) → Details View, **Dimensions:**, H → [input value, 15] (fig. c);

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.***

<u>*Reference thread contour generation (one pitch length)*</u>

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.

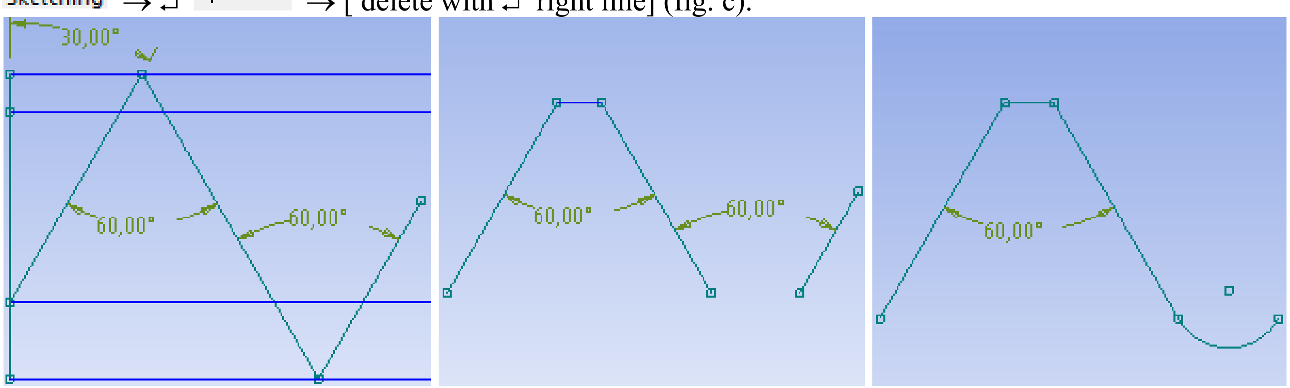
Generating profile lines

↲ 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.***

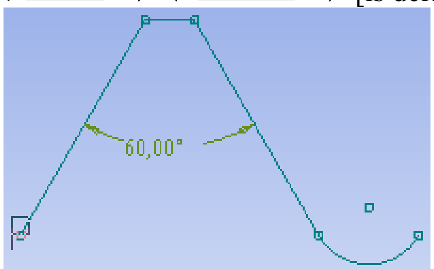
<u>*Generation of the screw thread by multiplying the reference contour*</u>

Multiply the reference contour

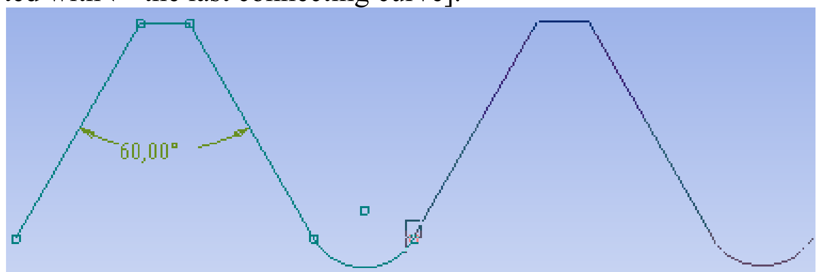
↲ Modify → ↲ Replicate → [value, 0] (appear: r = 0 ° f = 2) → [will be selected with ↳ a point in the graphics area] (context menu appears) → ↳ Selection Filter → ↲ 2D Edge → [will be selected with ↳ a point in the graphics area] (context menu appears) → ↲ End / Set Paste Handle → ↳ Selection Filter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

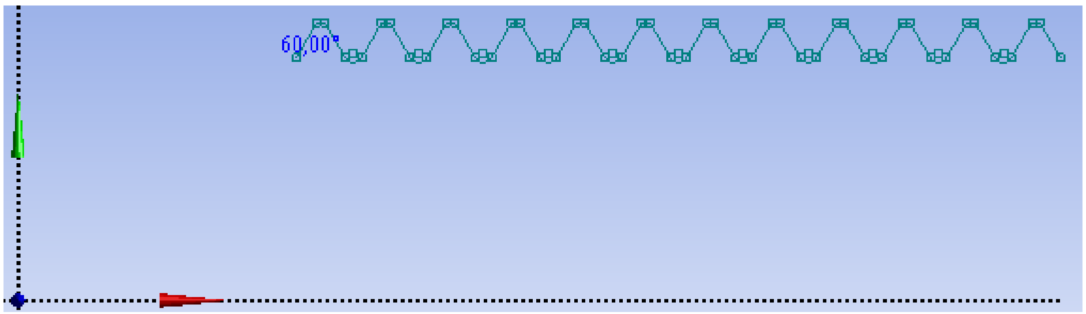
↲ Modify → ↲ Trim → [is deleted with ↲ the last connecting curve].



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

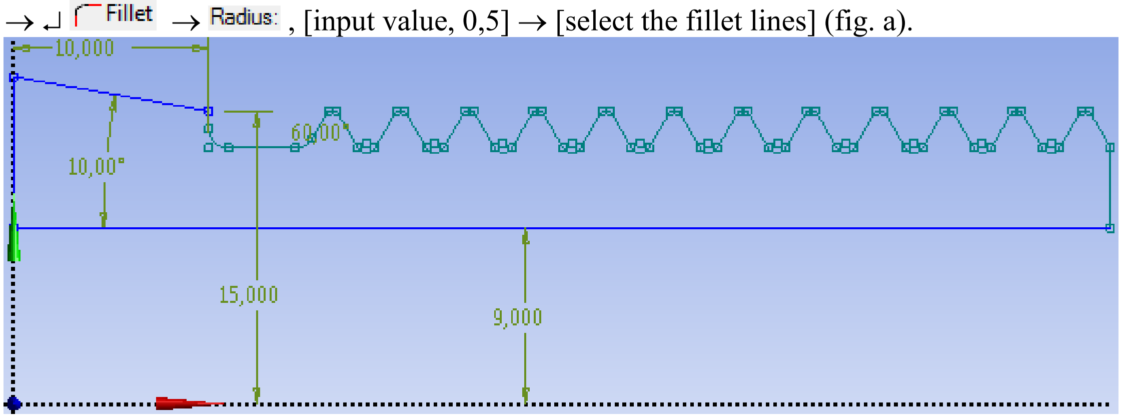
↲ Vertical → [select with ↲ the line parallel to the X axis and the X axis] (the dimension is automatically displayed) → Details View, **Dimensions:**: ↲ ☐ V → [input value, 9/15] (the sequence is repeated for the other lines, fig. a).

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

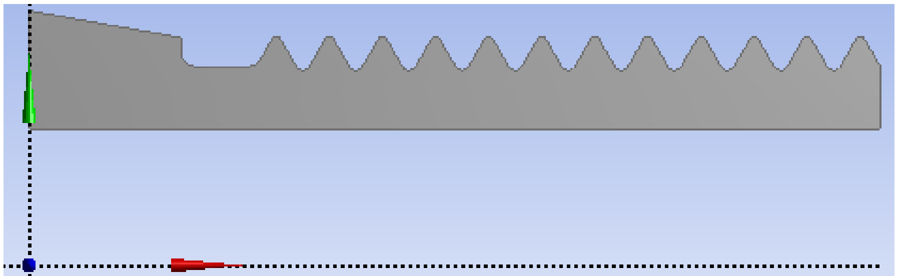
↲ Modify → ↲ Fillet → Radius: , [input value, 0,5] → [select the fillet lines] (fig. a).



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

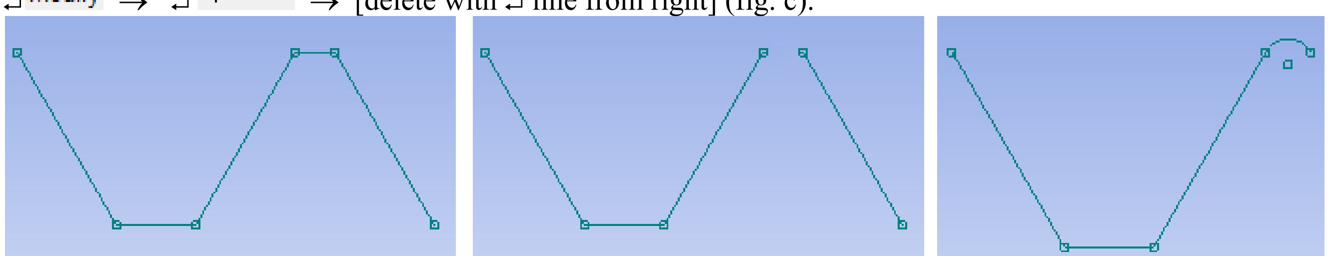
↵ Modify → ↵ Trim → [delete with ↵ the last one in the thread connection area] (fig. b).

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.***

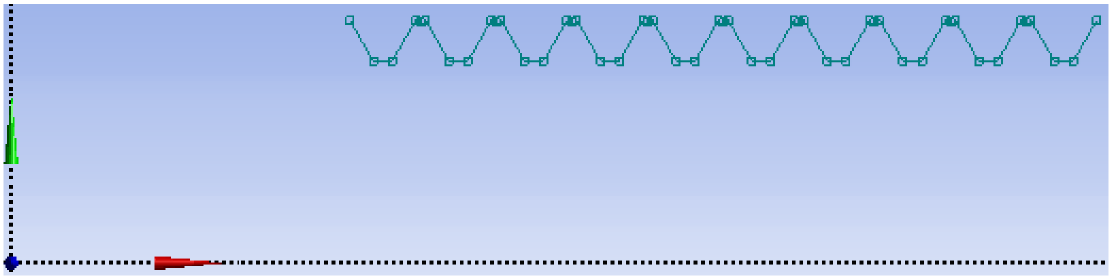
Generating nut thread by multiplying reference contour

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

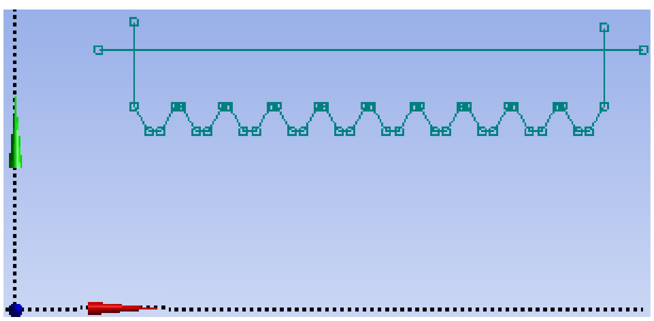
↵Draw → ↵ Line → [draw 2 vertical lines and one horizontal line with ↵ respecting the conditions of vertical and horizontal directions, respectively symbols V and H, respectively].

Delete lines

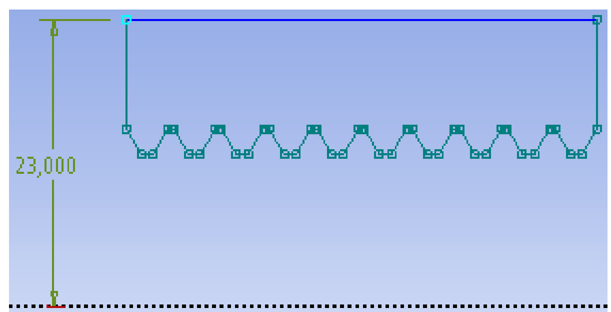
↵Modify → ↵ Trim → [delete with ↵ the ending line].

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).



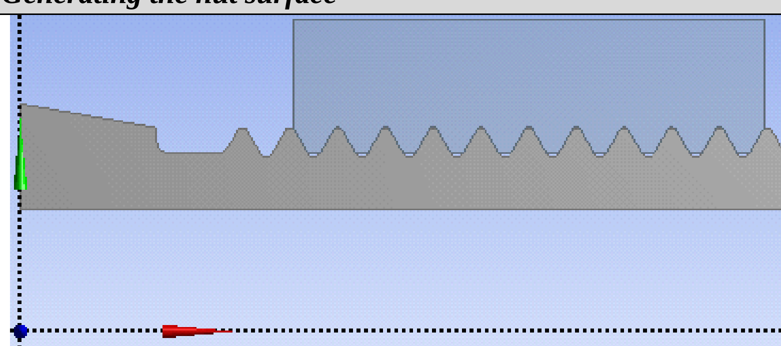
a.



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, Piuliţă].



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

, 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

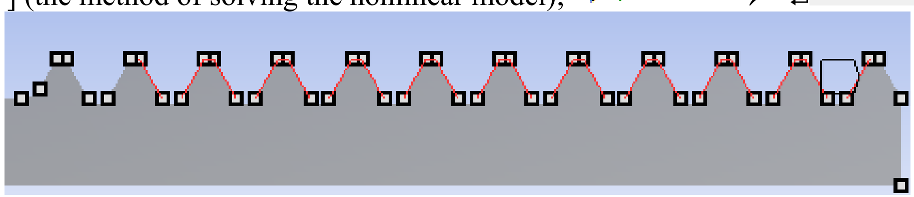
Outline: ↵ ⊞ Geometry → ↵ Surub → Details of "Surub", **Material**: ↵ Assignment → [select from the list ↵, ↵ Structural Steel (default)]; ↵ Piulita → Details of "Piulita", **Material**: ↵ Assignment → [select from the list ↵, ↵ Structural Steel (default)];

Setting the model type (axial asymmetric)

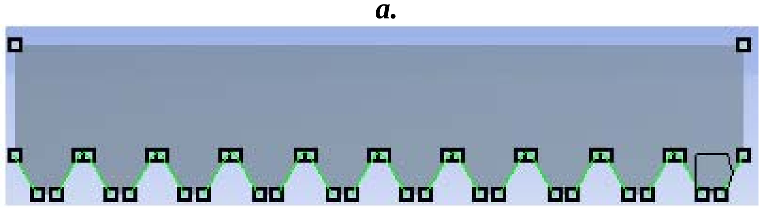
Outline: ↵ ⊞ Geometry → Details of "Geometry", **Definition**: ↵ 2D Behavior, [select from the list ↵, ↵ Axisymmetric].

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.

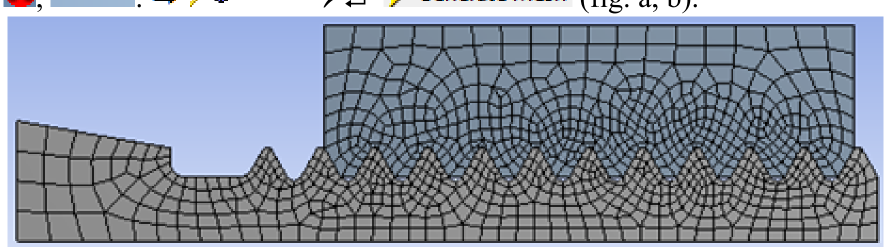


b.

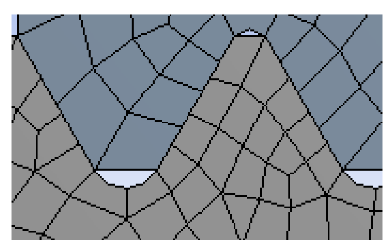
C.4.3 Model discretization and analysis type setting

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).

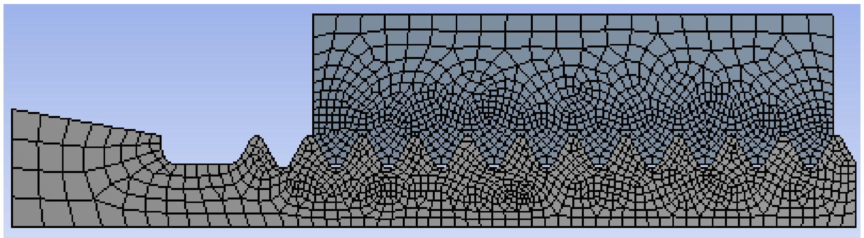
Setting finite element dimensions on contact surfaces

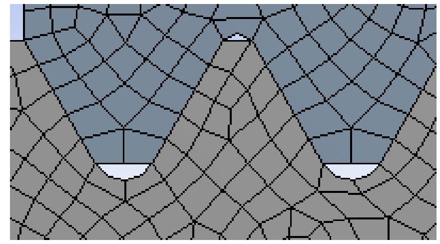
↵ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, **Scope**: ↵ Geometry → ↵ Piulita → ↵ Hide Body → ↵ → [select with Ctrl + ↵ the thread lines of the screw threads] → ↵ Apply; Details of "Edge Sizing" - Sizing: ↵ Element Size, [input value, 1,0] → ↵ Piulita → ↵ Show Body → ↵ Surub

→ ↵ 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).



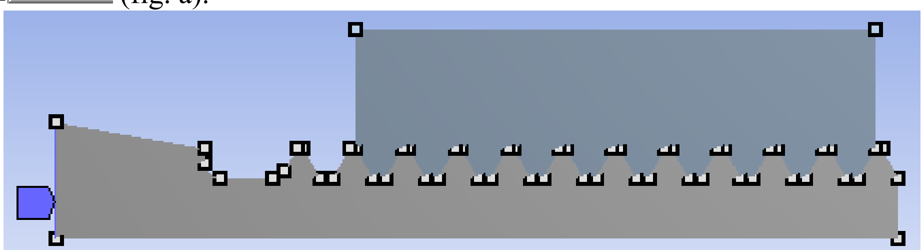


c. **d.**

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).

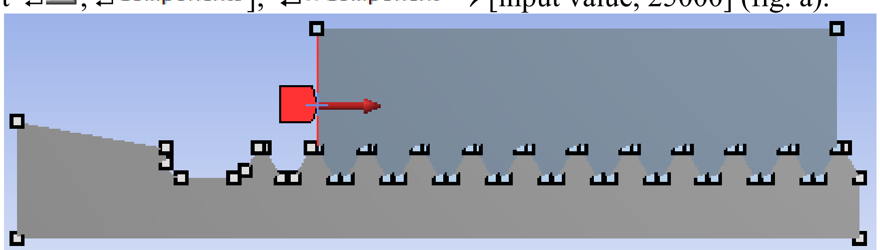


a.

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.

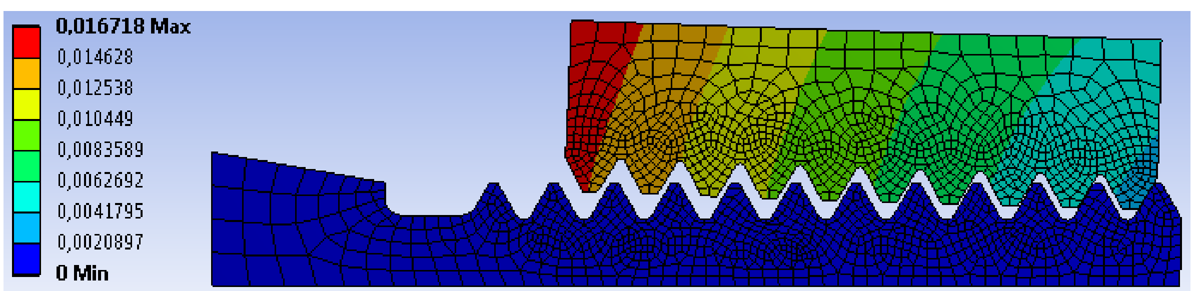
D.3 Launching the solving module

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

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

M, 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).

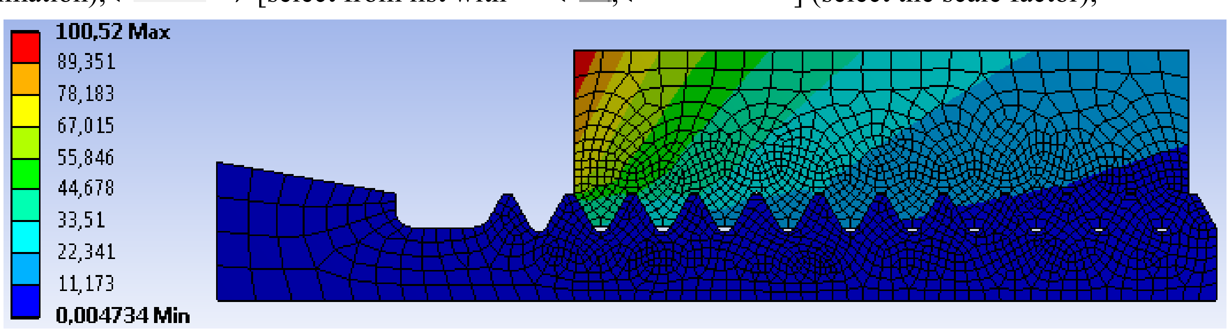


a.

E.2 Viewing the stresses field

Viewing the equivalent stress field

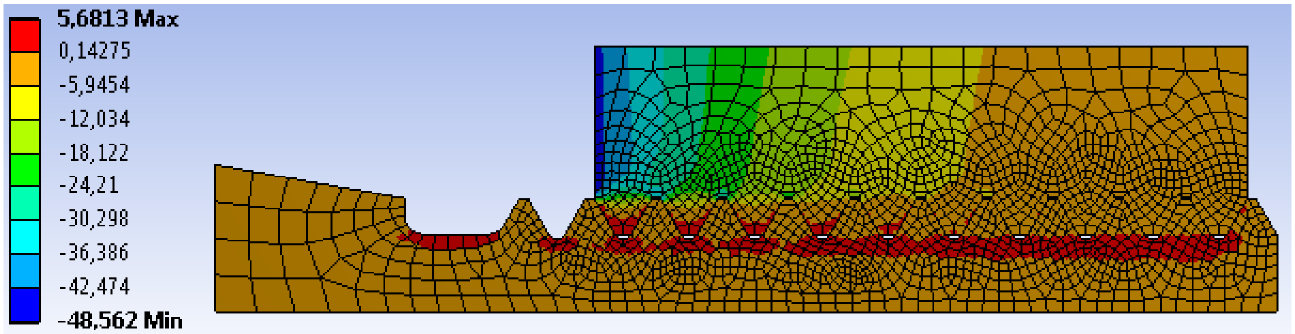
M, Outline: ↵⊞ Solution (A6) → ↵ Equivalent Stress (fig. a) ; Graph → ↵ Animation ▶ ■ (view the animation); ↵Result → [select from list with ↵▾, ↵1.0 (True Scale)] (select the scale factor);



a.

Viewing the axial stress field

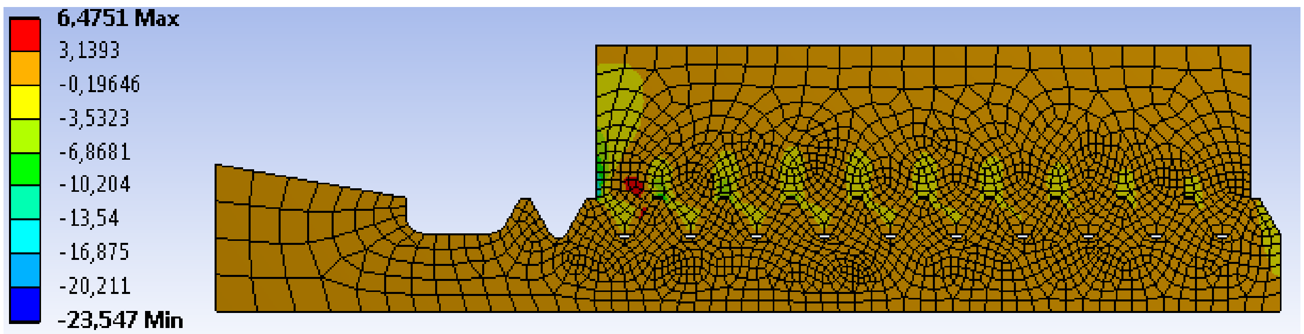
Outline: Solution (A6) Normal Stress (fig. b).



b.

Viewing the radial normal stress field

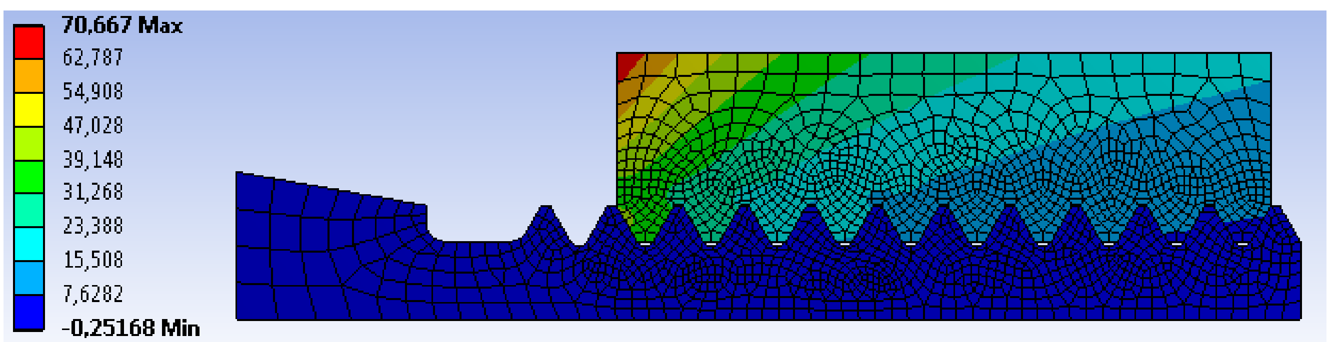
Outline: Solution (A6) Normal Stress 2 (fig. c).



c.

Viewing the tangential normal stress field

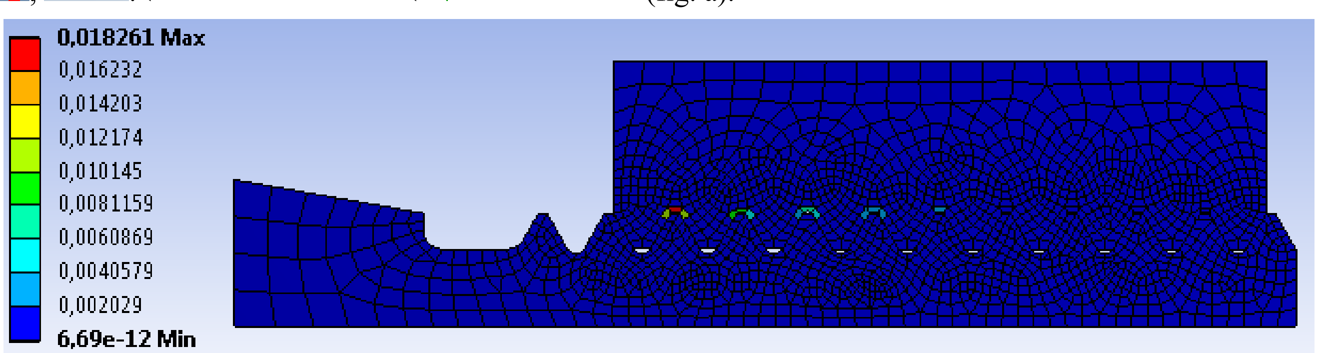
Outline: Solution (A6) Normal Stress 3 (fig. d).



d.

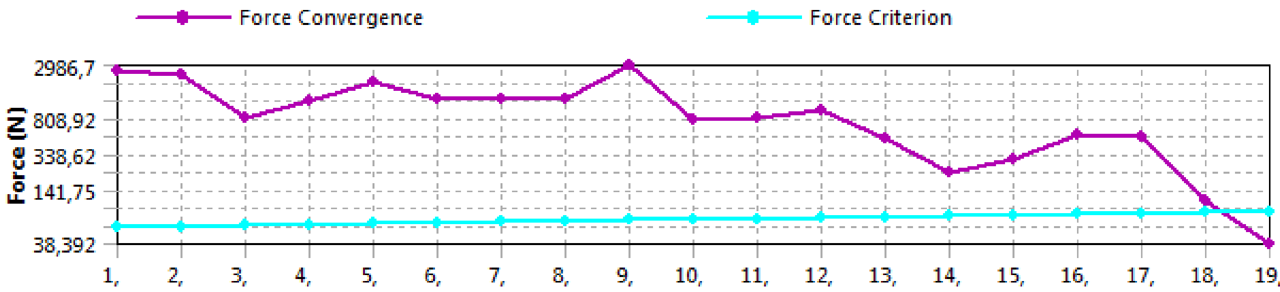
E.3 Viewing the structural error

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



a.

E.4 View the convergence graph of the solution of the nonlinear problem
, Outline: Solution Information (fig. a).

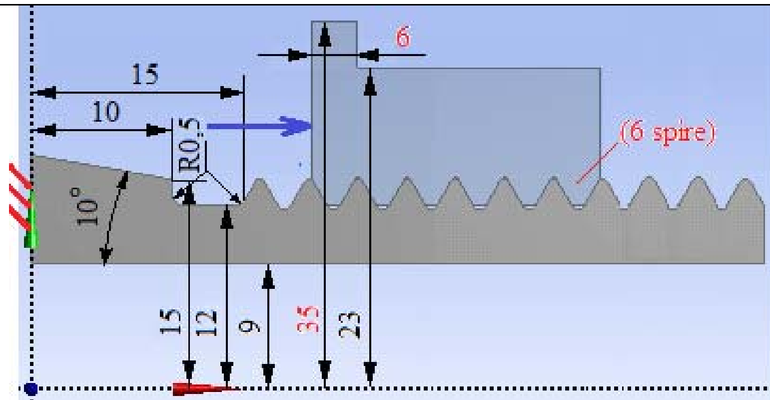


a.

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.



a.

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.