Application: AEF-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.

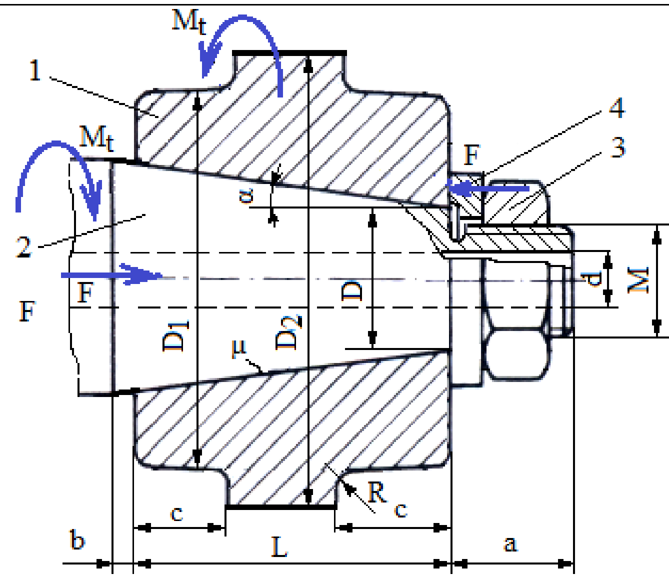
Tight assembly on the cone frequently used in the construction of mechanical systems form complex spatial structures involving mechanical frictional contacts that participate in load transmission. Starting from the fact that these structures cannot be accurately analyzed with classical theoretical and / 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,

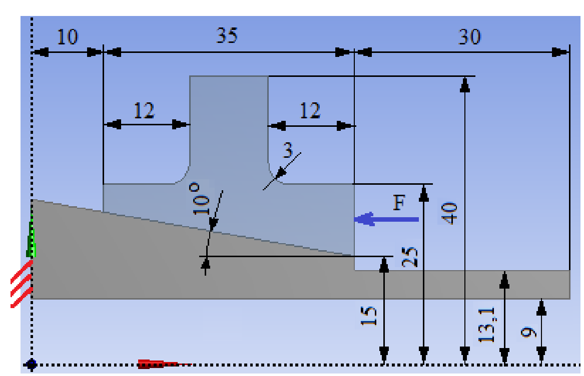
the deformation takes place statically (the variation of the deformation force over time is not taken into account).

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 $\mu = 0.2$.

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, $E = 206000 \text{ N / mm}^2$;

- 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

, Toolbox: ↵ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural în *AEF-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... → Save As, File name: [input name, *AEF-A.9*] → ↵ 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 / 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)

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

C3.2 Sketch generation 1 (shaft)

Viewing default plane (XY)

, 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

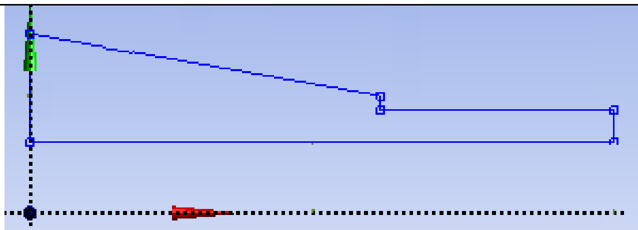
↵ Vertical → [select with ↵ the lines parallel to the X axis] (the dimension is automatically displayed) → Details View, **Dimensions:**: ↵ V → [input value, 15/13,1/9 fig. b).

Dimensioning the angles

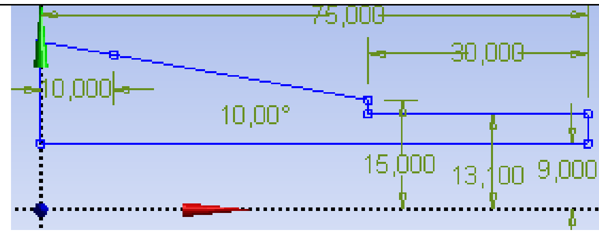
↵ Angle → [select with ↵ angle lines] (automatically view dimension) → Details View, **Dimensions:**: ↵ A → [input value, 10 fig. b).

Edit dimensions

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



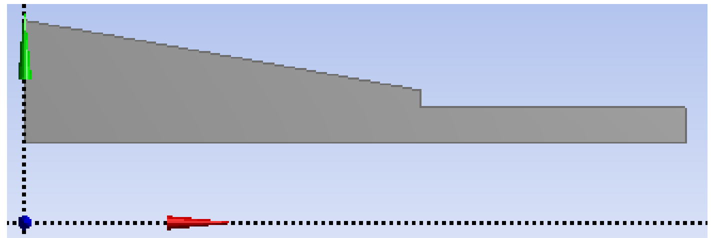
a.



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).

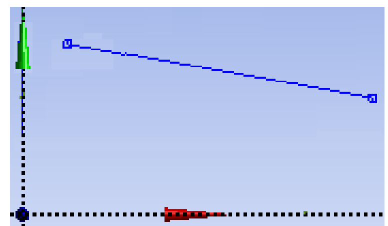
Contour generation

Activare schiță 1 / 1st sketch activation

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

Common line generation

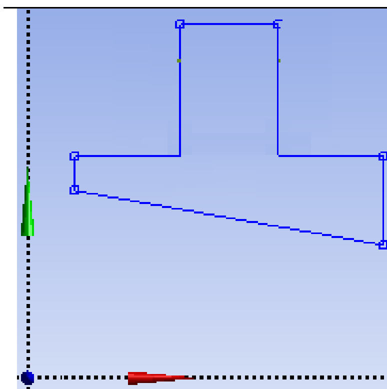
↵Sketching → Sketching Toolboxes: ↵Draw → ↵ Line → [select with ↵ the end points of the common line respecting the coincidence conditions P] (fig. a).



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).

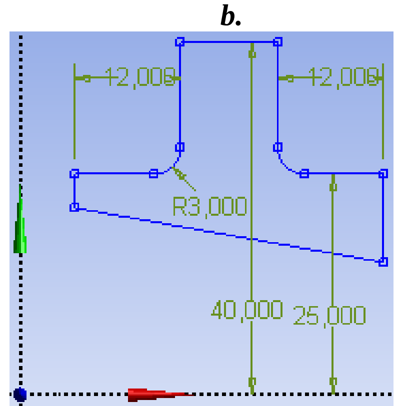
Contour dimensioning

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

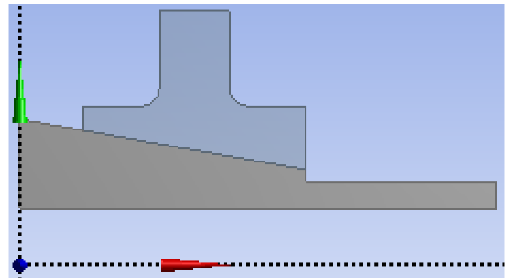
↵ Vertical → [select with ↵ the lines parallel to the X axis] (the dimension is automatically displayed) → Details View, Dimensions:: ↵☐ V → [input value, 40/25] (fig. c).



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

, Project Schematic: ↳ Model → ↵ 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

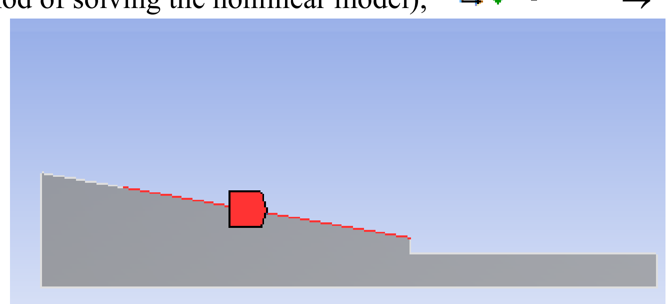
Outline: ↵ Geometry → ↵ Surub → Details of "Surub", Material : ↵ Assignment → [select from the list ↵ , ↵ Structural Steel (default)]; ↵ Piulita → Details of "Piulita", Material : ↵ Assignment → [select from the list ↵ , ↵ Structural Steel (default)];

Setting the model type (axial asymmetric)

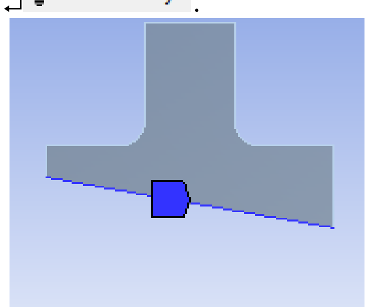
Outline: ↵ Geometry → Details of "Geometry", Definition: ↵ 2D Behavior, [select from the list ↵ , ↵ Axisymmetric].

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.

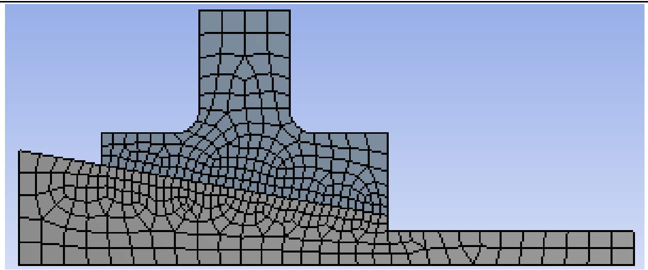


b.

C.4.3 Model discretization and analysis type setting

Automatic meshing

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



a.

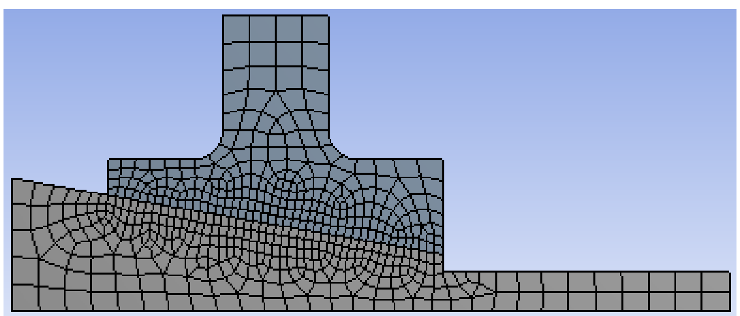
Obs. In fig. a, there are discontinuities of the finite element structure at the level of the contact surface, which leads to an inadequate modeling of the contact phenomena and it is necessary to correct the discretization ensuring the continuity at the nodal level.

Setting finite element dimensions on contact surfaces

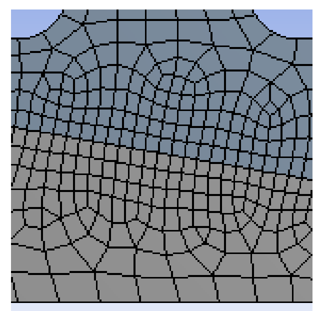
↳ Mesh → ↲Insert → ↲ Sizing → Details of "Sizing" - Sizing, Scope: ↲Geometry → ↳ Butuc → ↲ Hide Body → ↲ → [select with Ctrl + ↲ the contact line] → ↲ Apply; Details of "Edge Sizing" - Sizing: ↲ Element Size, [input value, 0,8] → ↳ Butuc → ↲ Show Body → ↳ Arbore → ↲ Hide Body → ↲ → [select with Ctrl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



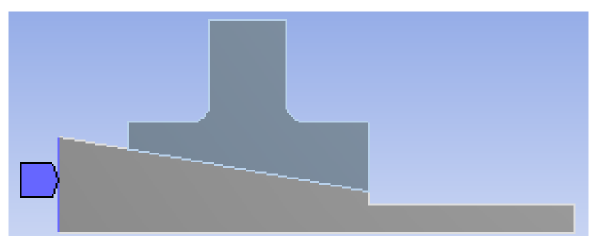
b.



c.

C.5. *Supports and restraints modelling*

Fixed support constraint (cancels all 6 degrees of mobility)

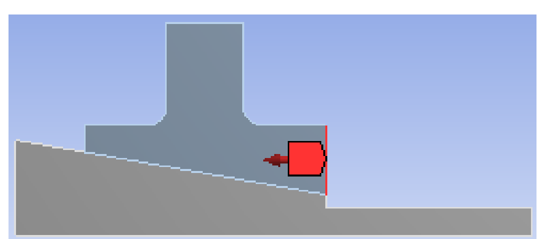


a.

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).

C.6 Load modeling

Distributed force load on one edge



a.

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).

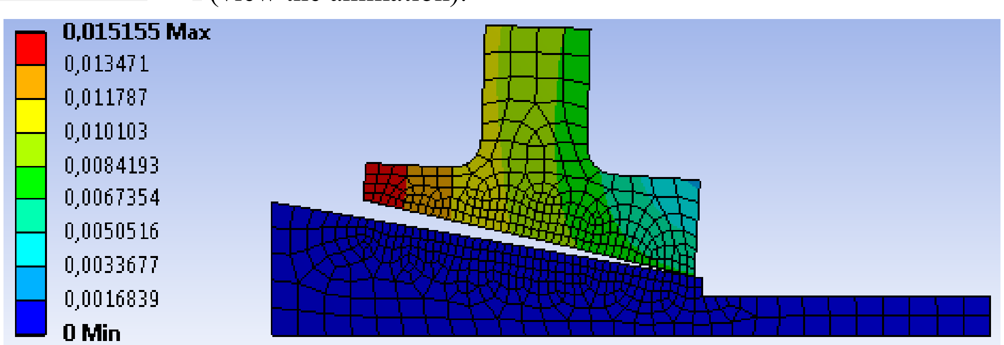
D. SOLVING THE FEA MODEL

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
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 ↵, ↵ 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).

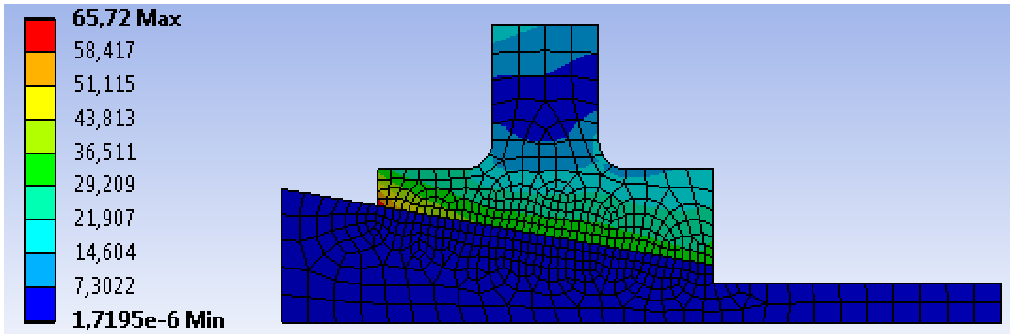


a.

E.2 Viewing the stresses field

Viewing the equivalent stress field

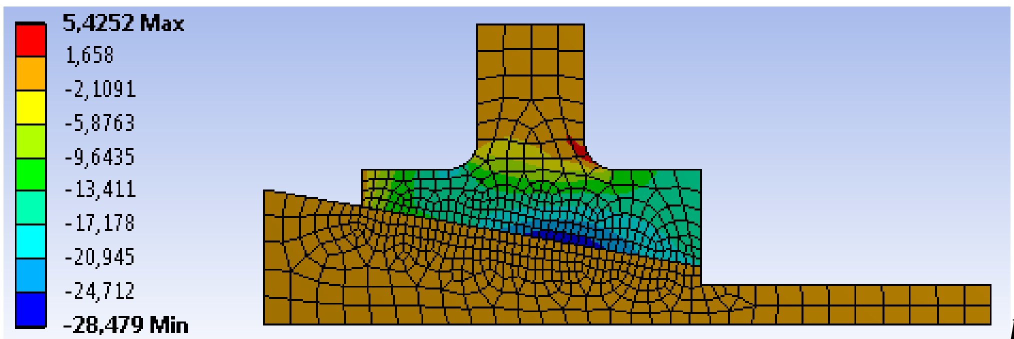
, Outline: ↵ ⊞ ? Solution (A6) → ↵ Equivalent Stress (fig. a) ; Graph → ↵ Animation (view the animation); ↵ Result → [select from list with ↵ ▼, ↵ 1.0 (True Scale)] (select the scale factor);



a.

Viewing the axial stress field

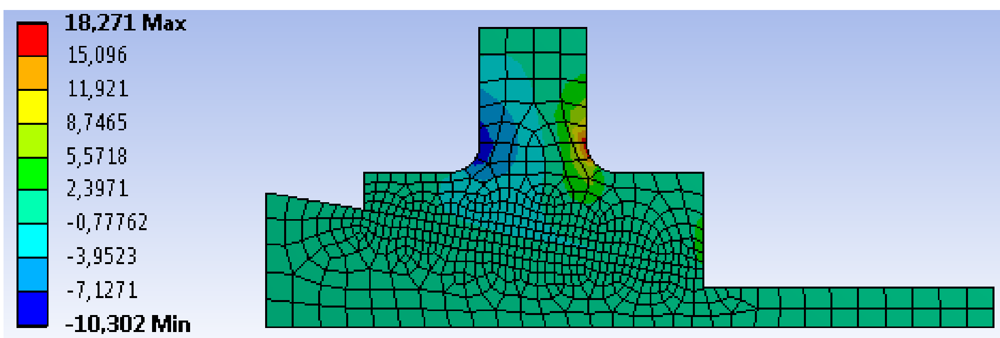
, Outline: ↵ ⊞ ? Solution (A6) → ↵ Normal Stress (fig. b).



b.

Viewing the radial normal stress field

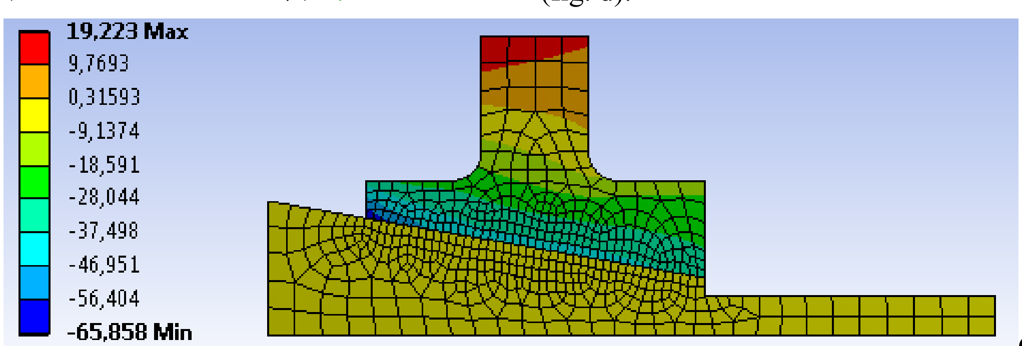
, Outline: ↵ ⊞ ? Solution (A6) → ↵ Normal Stress 2 (fig. c).



c.

Viewing the tangential normal stress field

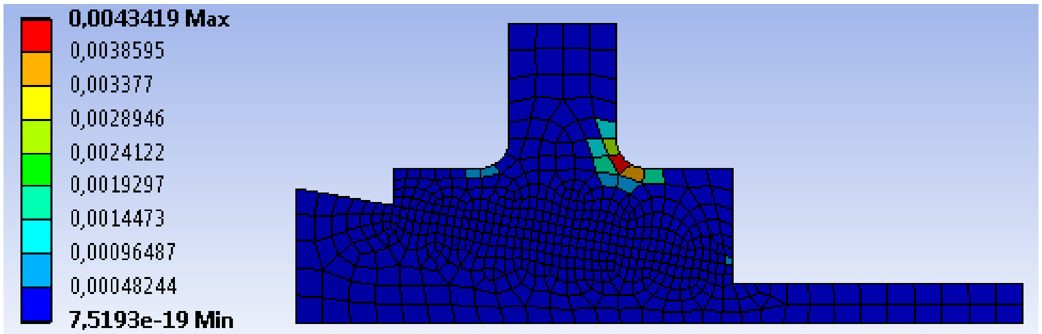
, Outline: ↵ ⊞ ? Solution (A6) → ↵ Normal Stress 3 (fig. d).



d.

E.3 Viewing the structural error

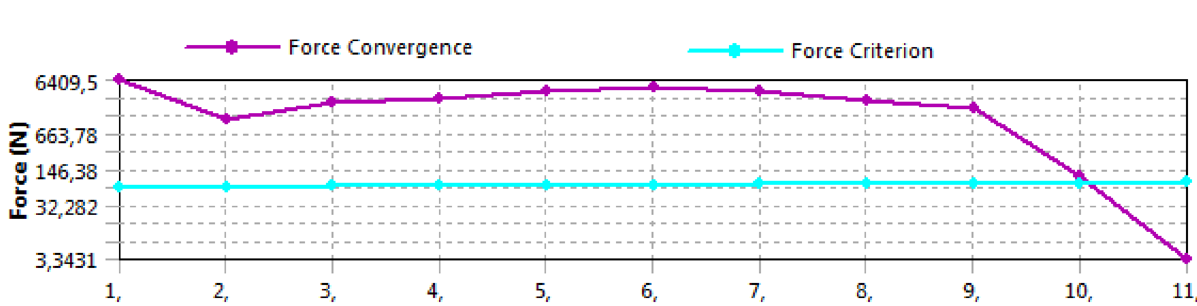
Outline : Solution (A6) Structural Error (fig. a).



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

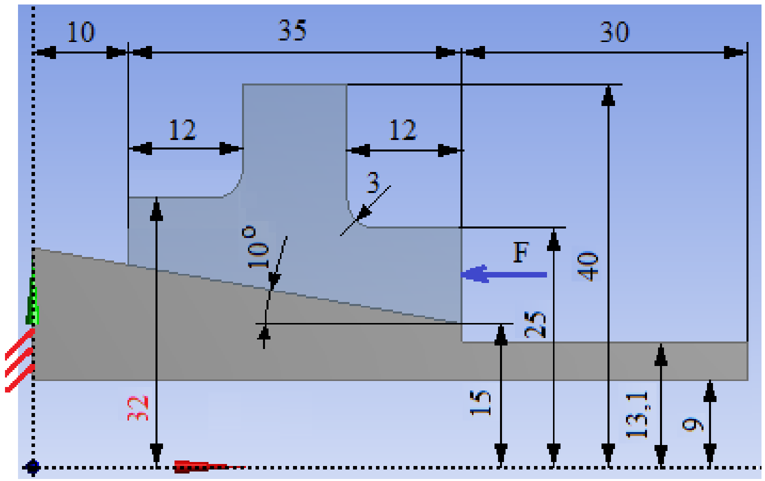
The much reduced values of the structural error field (max 0.0436 mJ, subchapter E.3) indicate that the stress values are close to the exact ones. In addition, from subchapter. E.4 highlights the fast convergence (19 pitches) of the model solving algorithm and the calculation time is reduced.

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; M, 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.