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
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

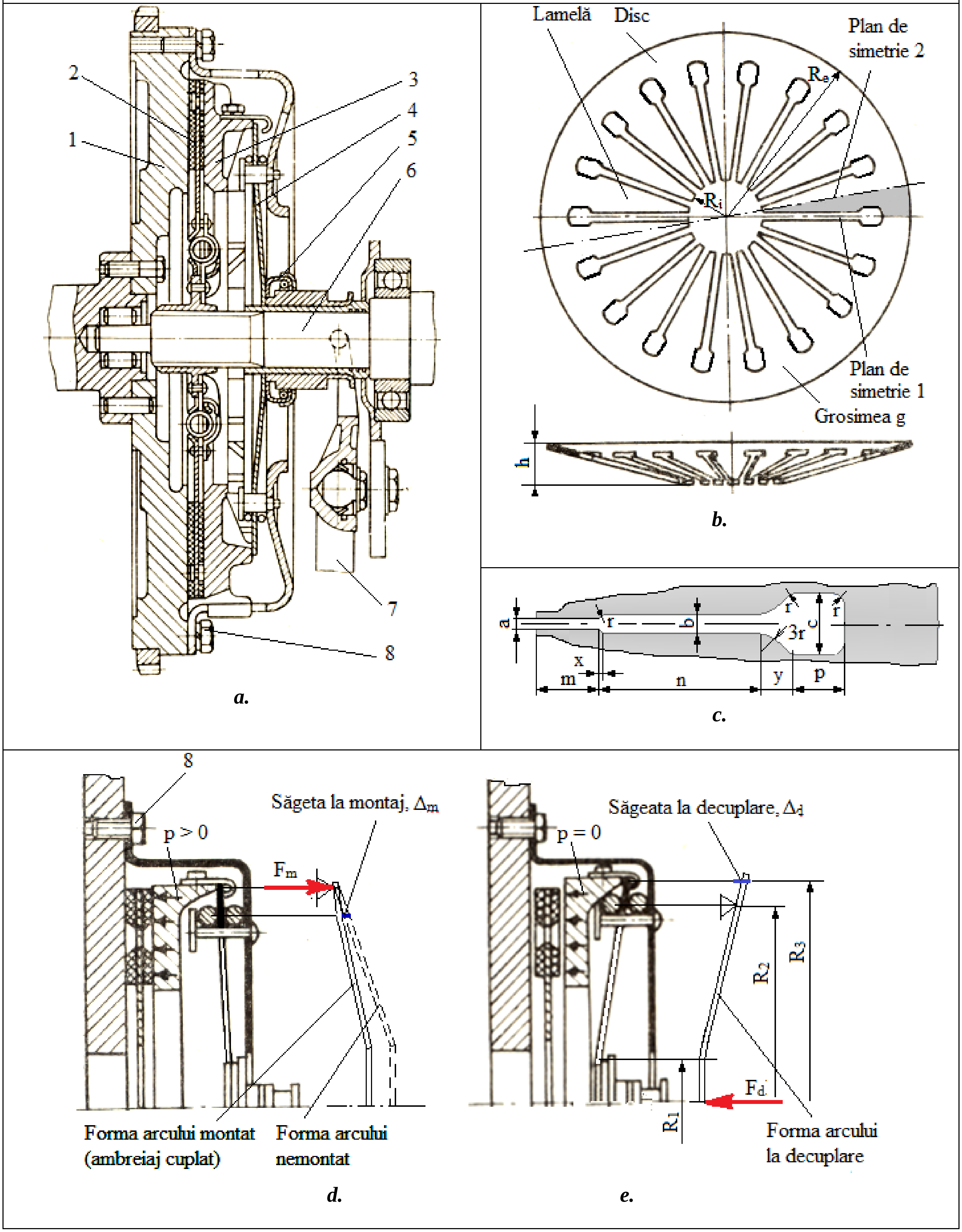
The clutch of the car is a normal intermittent mechanical coupling coupled *with the main function of decoupling-coupling* the transmission of the car in case moment of changing gears or brakes under load. In addition, secondary functions are required for optimum operation: *smooth decoupling and coupling, 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.



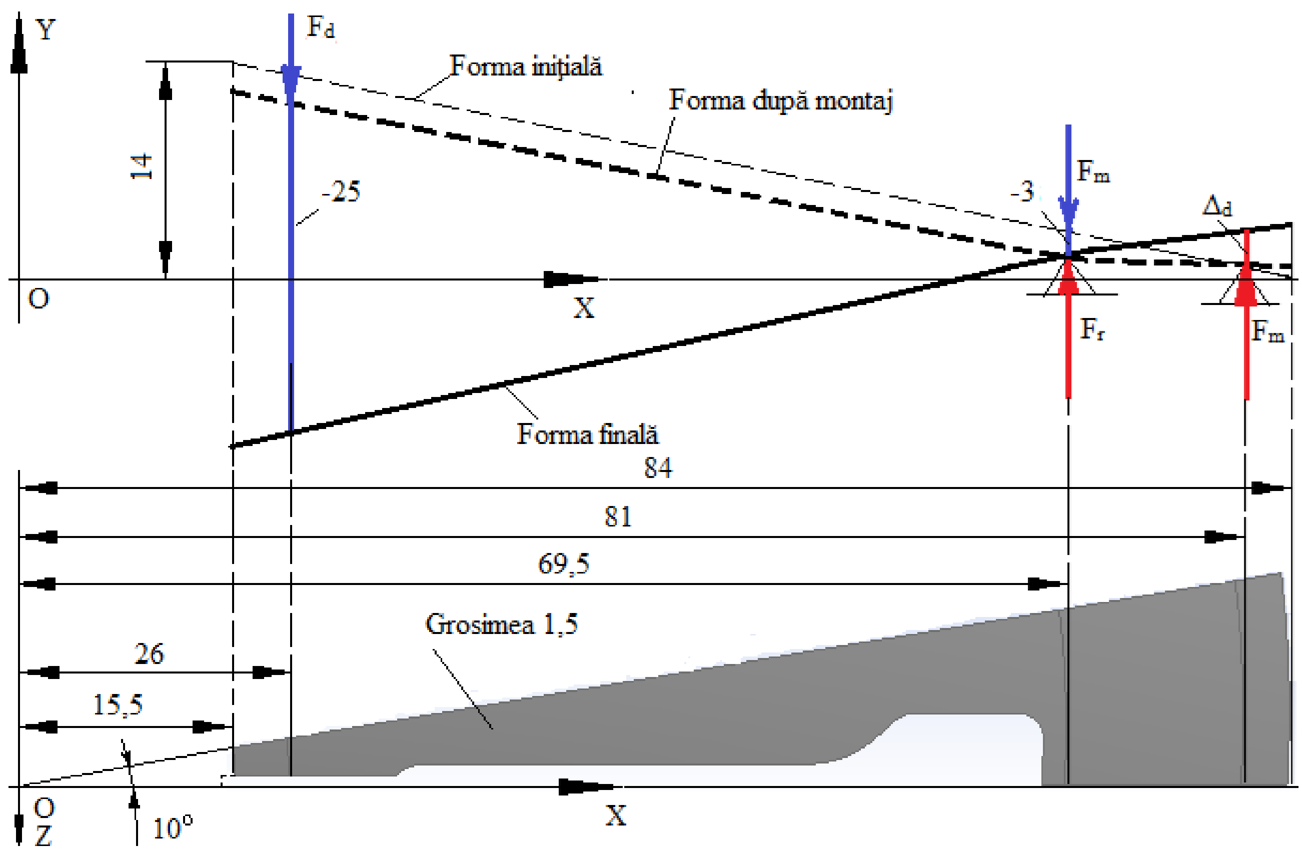
A.2. Application description
Structure and operation of the clutch with diaphragm spring. The mechanical clutch in the figure above transmits the torque by friction from the flywheel assembly 1 and the pressure plate 3 to the disc 2 and through the groove to the main shaft of the gearbox 6. This process 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

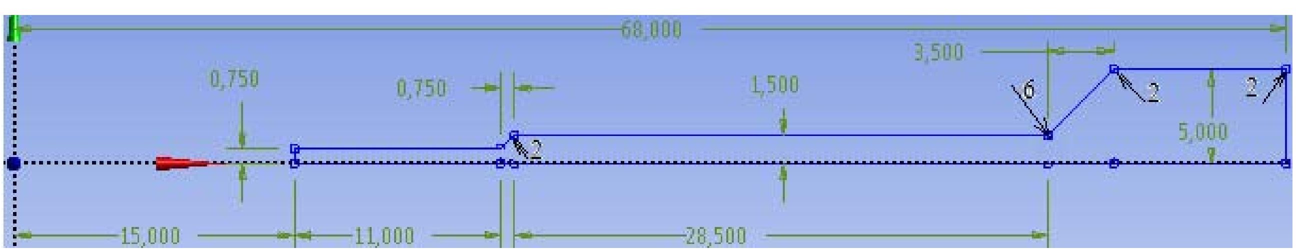
B.1. The model definition
Since the diaphragm spring has a reduced thickness (1.5 mm), the variations of the unknown internal parameters (displacements, deformations and stresses) are insignificant in the normal direction at the surface and a *2D model* is adopted for analysis. On the other hand, the 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 2D finite elements. 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

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... → Save As, File name: [enter name, FEA-A.5] → ↵ 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 / value, 206000] → ↵ Update Project → ↵ Return to Project (others parameters are default).

C.3 Geometric modelling

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

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].

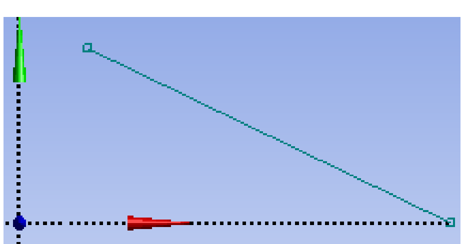
Dimensioning of reference line

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

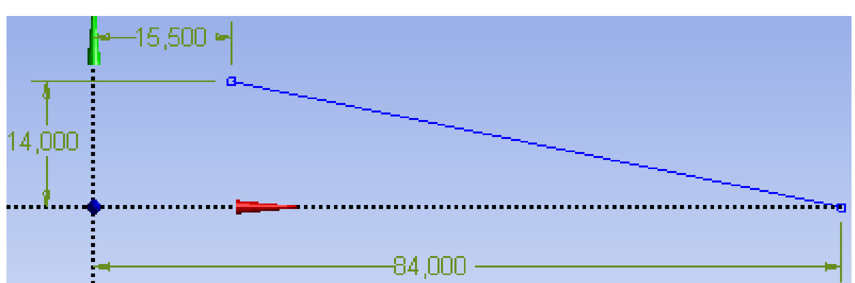
↵ Horizontal → [selecting succesive ↵ the pair: point on the reference line and the OY axis, the dimension appears (fig. b)] → Details View, Dimensions: 1: ☐ H → [input value, 15,5/84,0].

↵ 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.

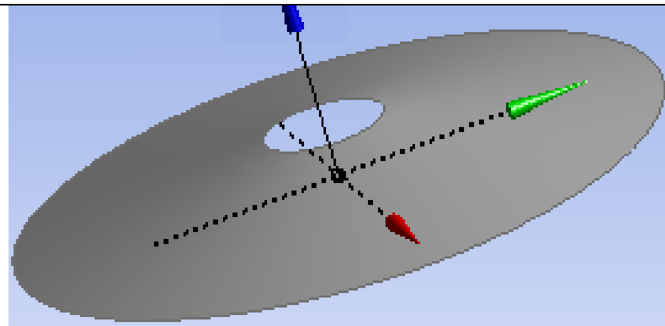


b.

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 ; DM: ↵ (visualization of the coordinate system attached to the geometric model).



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

Generating sketch helplines

DM, 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

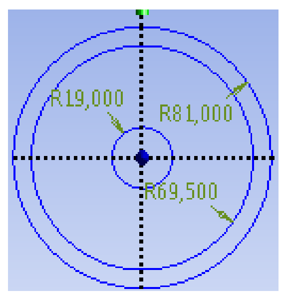
↵ Draw → ↵ Circle → [the circular line is generated by selecting with ↵ the center of the circle in the center of the coordinate system (coincidence symbol P appears), moving in radial direction and release ↵ on the contour, fig. a] (this sequence is performed three times for each circle).

Dimensions helplines

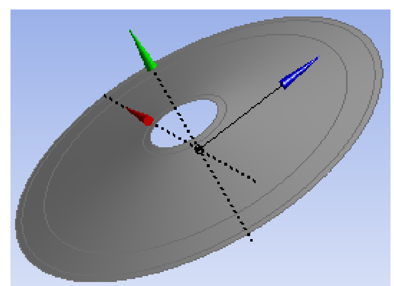
Tree Outline: ↵ Modeling → ↵ Sketch2 ; ↵ Dimensions → ↵ Radius → [select with ↵ the circular line, the dimension is automatically displayed] → Details View, Dimensions: 1 : R → [input values: 19; 69,5; 81, (fig. a)].

Printing helplines on the reference surface

DM: ↵ 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

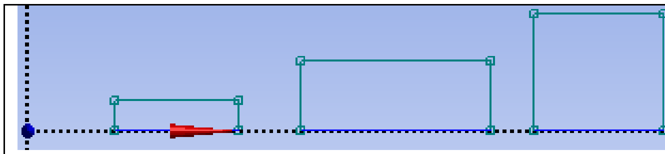
DM, Tree Outline: ↵ ZXPlane → ↵ Sketching → ↵ (New Sketch) [the sketch name is automatically indexed, Sketch3] → Details View , Details of Sketch3 : [change name Sketch3 name of the sketch, in Decupaj); ↵ (Look At Face/Plane/Sketch) [the selected plan will be automatically viewed, ZX]; ↵ (geometric model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

Generating rectangular lines

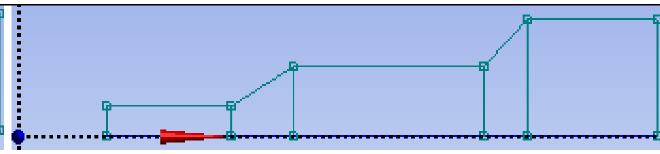
Tree Outline: ↵ Sketching → ↵ Draw → ↵ Rectangle → [select with ↵ the corner of the rectangular line at a point on the axis OX (coincidence symbol C appears), move the indicator and release ↵ in the other corner] (this sequence is carried out three times, fig. a).

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).



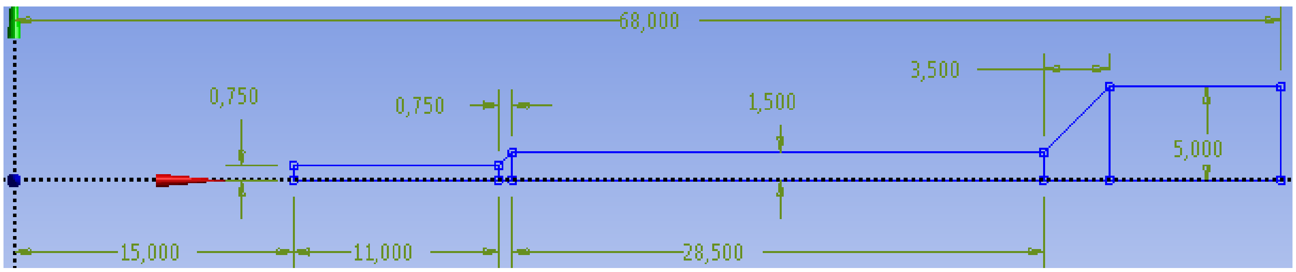
a.



b.

Dimensioning the sketch

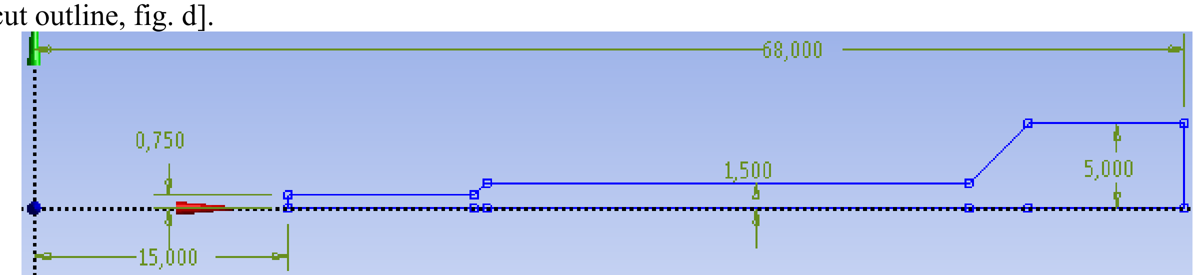
Sketching Toolboxes: ↵ Dimensions → ↵ Horizontal → [two entities (point, lines, axis) are selected with ↵ and the dimension appears automatically] → Details View, Dimensions: 1 : H → [input value (fig. c)];
↵ Vertical → [two entities (lines) are selected with ↵ and the dimension appears automatically]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



c.

Erasing lines

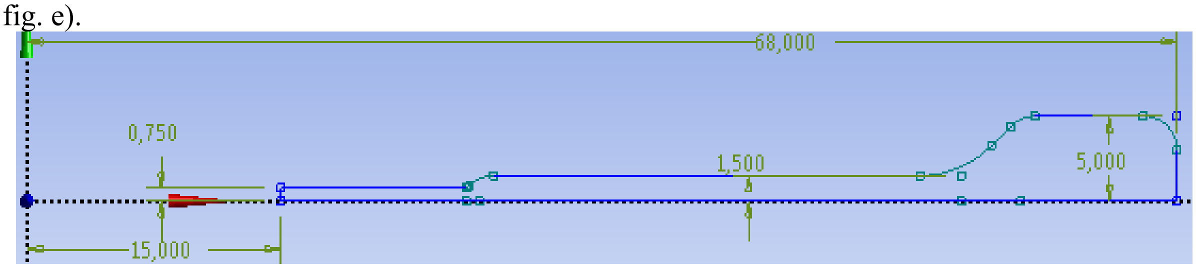
Sketching Toolboxes: →↵ Modify → ↵ Trim → [is deleted by selecting with ↵ lines that do not belong to the cut outline, fig. d].



d.

Generating radius

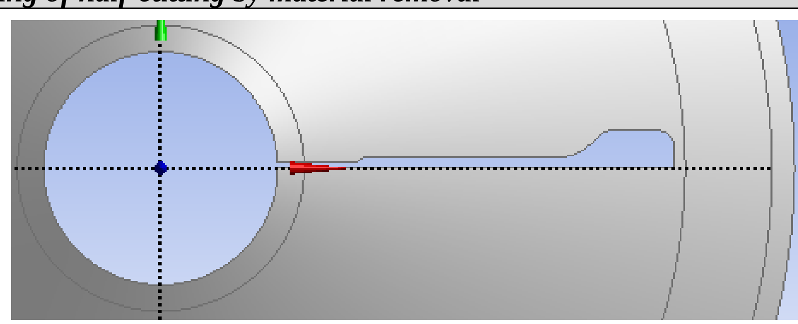
Sketching Toolboxes, Modify: → ↵ Fillet → [input in Radius: value, 2/6] → [the intersection point of the two connected lines is selected with ↵ and the connection line is automatically displayed] (the sequence is repeated 3 times for the connections with the radius of 2 mm and once for the connection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



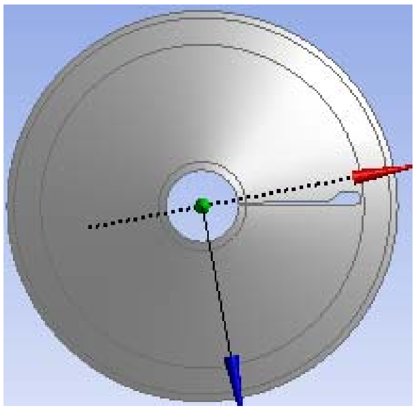
C.3.7 Model generation using cyclic symmetry

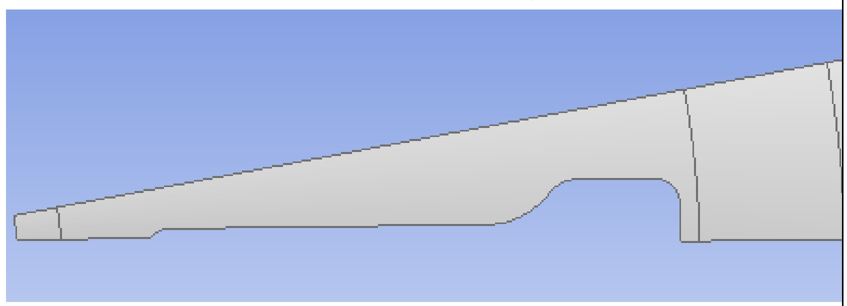
Generating of the symmetry coordinate system

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) → ↵ X (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

Λ , 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 .

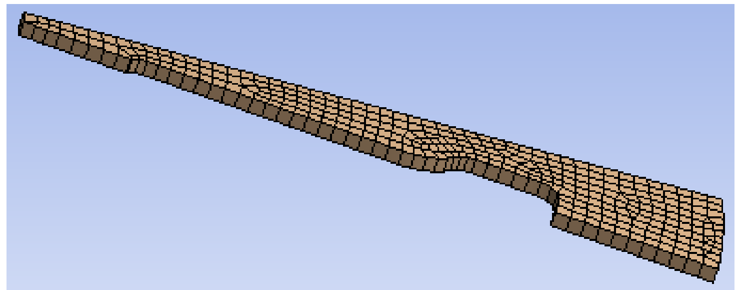
Setting the analysis parameters

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).

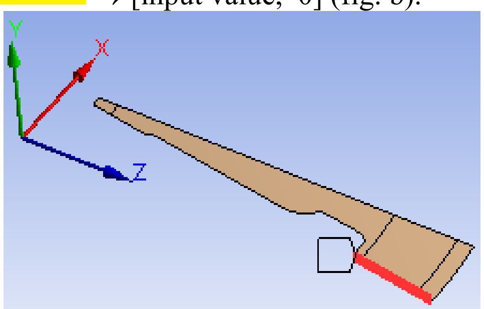


C.4.3 Supports and restraints modelling

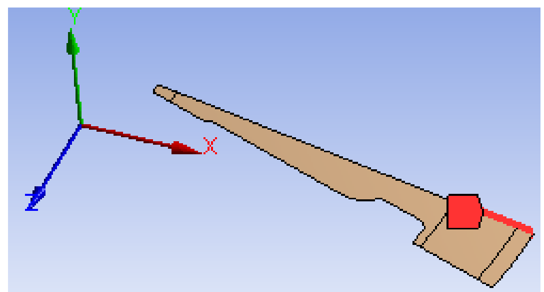
Generating of tangential (circumferential) displacement constraints associated with radial lines (ensures the behavior of the reduced symmetry model as well as the extended model)

, 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.

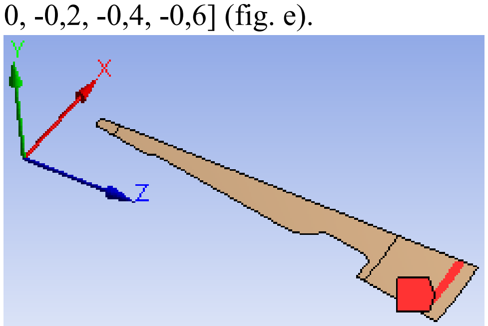
Case I (assembly model)

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

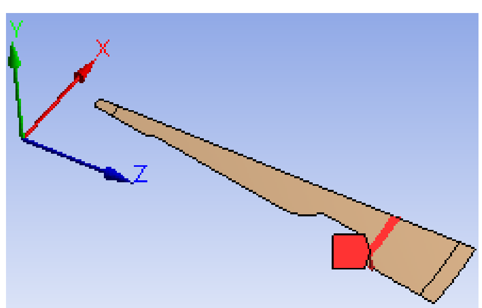
↲ ⊟ Static Structural (A5) → ↲ Supports ▾ → ↲ Displacement ; → ↲ → [selects the contact edge with the pressure disc]; Details of "Fixed Support": ⊟ Scope : ↲ Geometry → ↲ No Selection → ↲ Apply , Definition : ↲ Y Component → [input value, 0] (fig. c).

Generating of constraints with imposed axial displacements (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

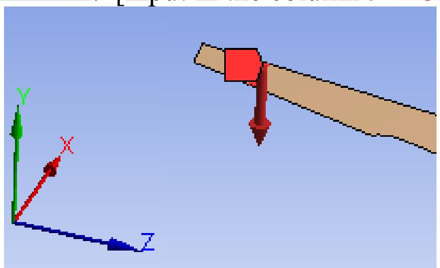
	Steps	Time [s]	☑ Y [mm]
1	1	0,	0,
2	1	1,	-0,2
3	2	2,	-0,4
4	3	3,	-0,6
*			

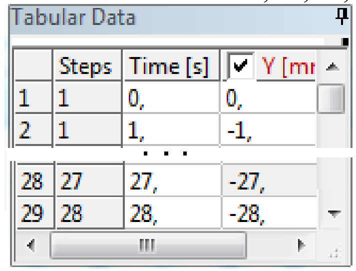
e.

Case II (decoupling model)

Generating of constraints with imposed axial displacements (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

Selecting the total displacements

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

Selecting the equivalent stress

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

Setting the circumferential stress *(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].

Setting the normal stress along the Z axis of the generated cylindrical coordinate system

↳ ⊞ 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].

Setting the structural error

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

Reaction force setting *(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

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

E. POST-PROCESSING OF RESULTS

E.1. Viewing the displacement field

Ⓜ, Outline: ↲⊞ Solution (A6) → ↲ Total Deformation .

↲ ISO (axonometric view).

↲ → [selecting from the list ↲▼, ↲ Smooth Contours] (visualization of smooth contours).

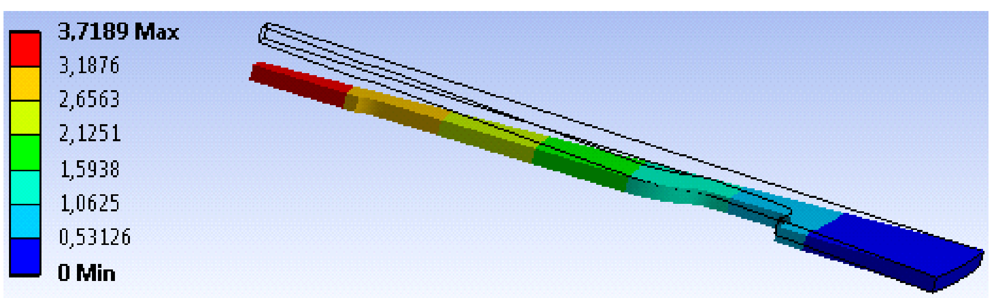
↲ → [selecting from the list ↲▼, ↲ Show Undeformed WireFrame] (visualization of the non-deformed wireframe structure).

↲Result → [selecting from the list ↲▼, ↲1.0 (True Scale)] (selecting displacement scaling).

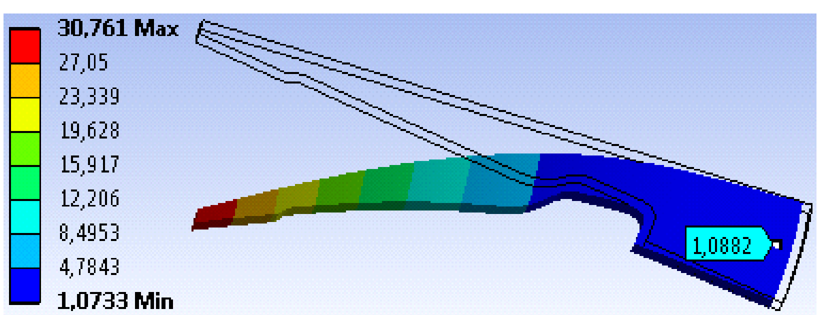
↲Probe → [is marked with ↲ a point in the area of the contact line with the pressure plate] (visualizing the value in a node).

Graph → ↲ Animation ▶ ■ (animating view).

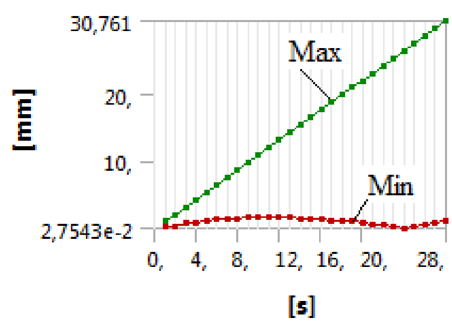
Cazul I / Case I



Cazul II / Case II



a.

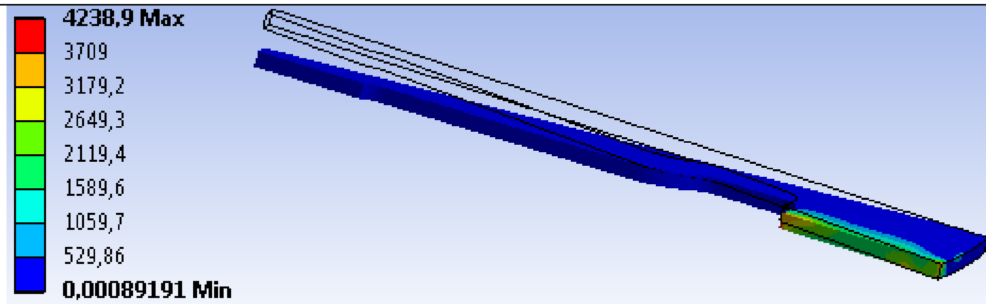


b.

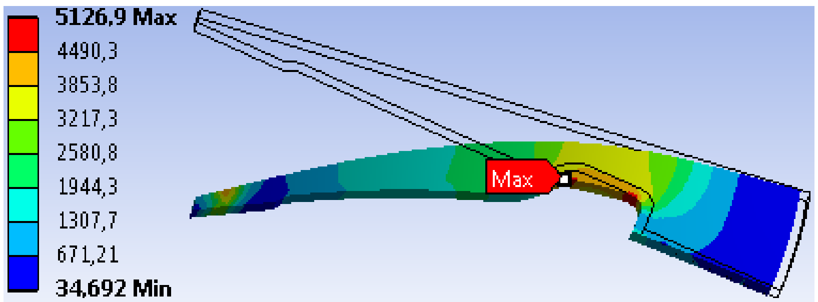
E.2 Viewing the equivalent stress field (von Mises)

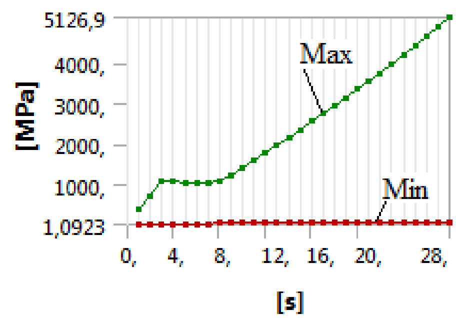
Outline: ↲⊞ Solution (A6) → ↲ Equivalent Stress .

Case I



Case II



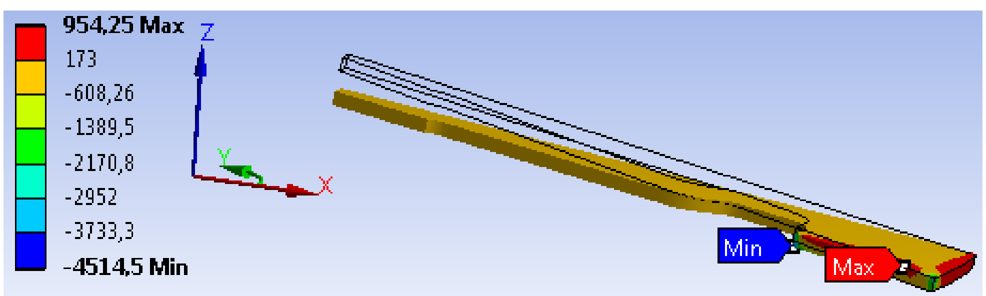


a. *b.*

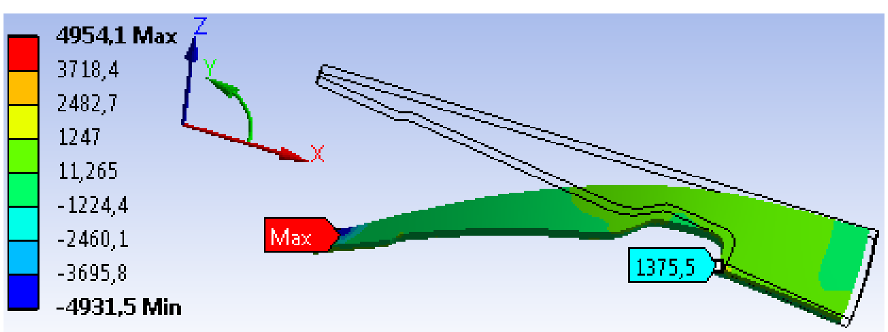
E.3 Viewing the circumferential stress field (radially normal)

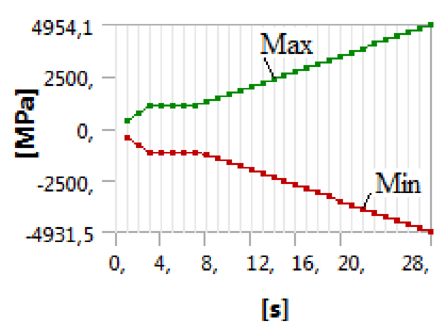
Outline: ↵ Solution (A6) → ↵ Normal Stress; ↵ (measurement of the node with the maximum circumferential stress); ↵ (measurement of the node with the minimum circumferential stress).

Case I



Case II



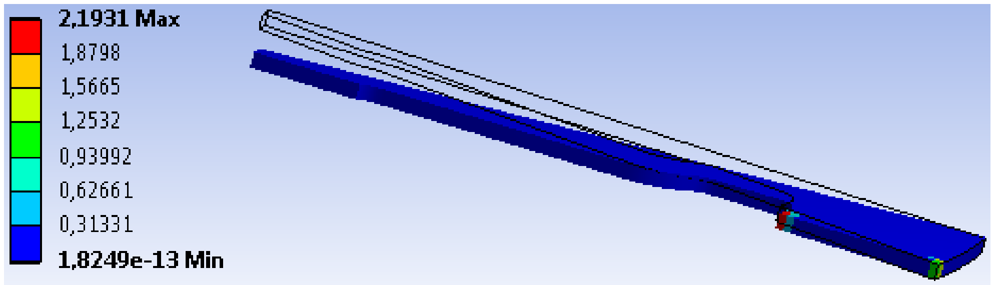


a. *b.*

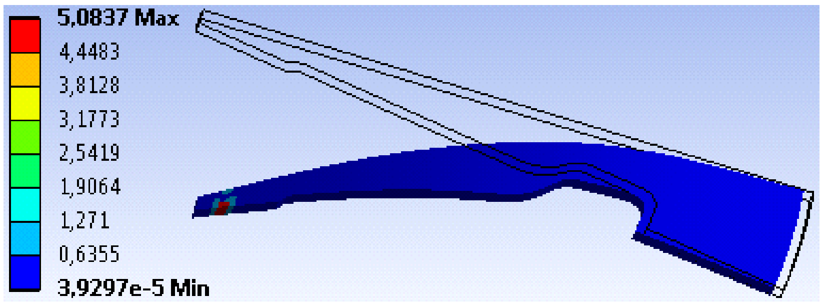
E.4 Visualization of the structural error field

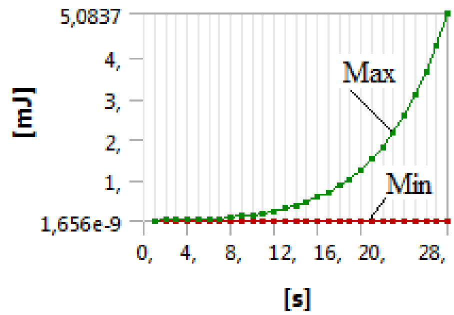
Outline: ↵ Solution (A6) → ↵ Structural Error .

Case I



Case II



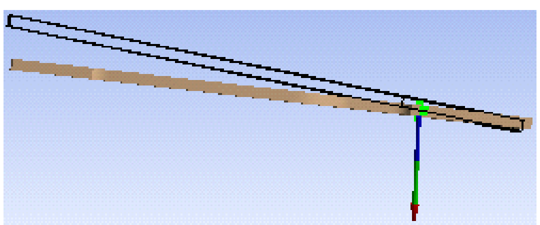


a. *b.*

E.5 Viewing the reaction forces

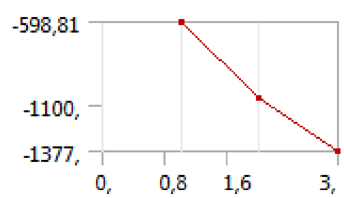
Outline: ↵ Solution (A6) → ↵ Force Reaction; [activation of the context menu with ↳ in the graphics area] → ↵View → Left (projection visualization, fig. a); ↵Graph → [the reaction force graph is automatically displayed according to the loading steps), fig. c]; ↵ Animation (viewing the animation).

Case I



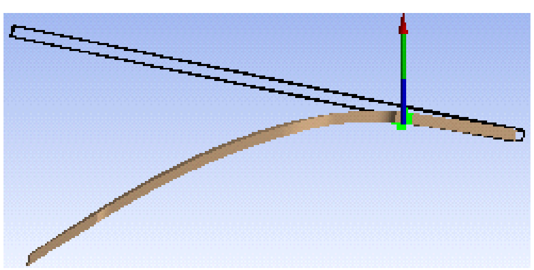
Tabular Data

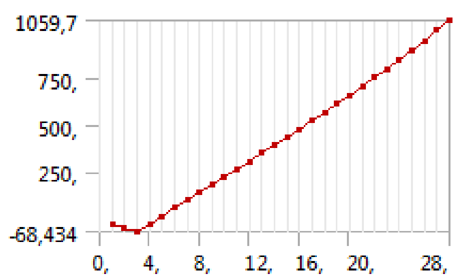
	Time [s]	Force Reaction (Y) [N]
1	1,	-598,81
2	2,	-1054,6
3	3,	-1377,



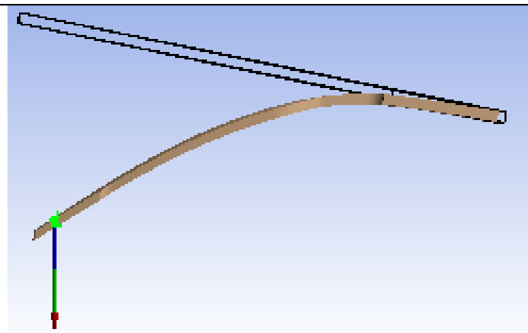
a. *b.* *c.*

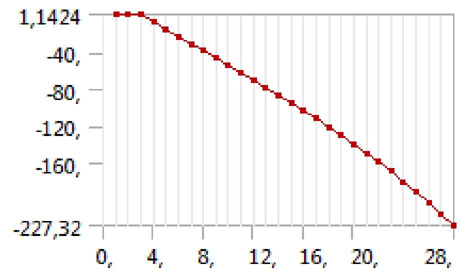
Case II





a. *b.*

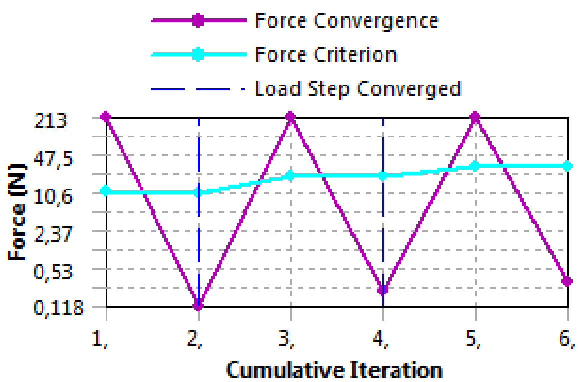




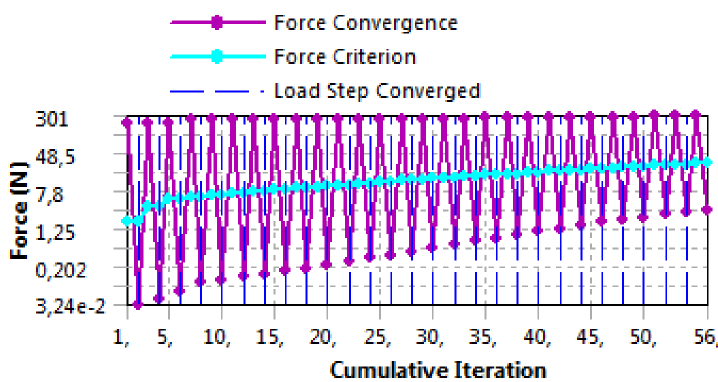
c. *d.*

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.