Application: AEF-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

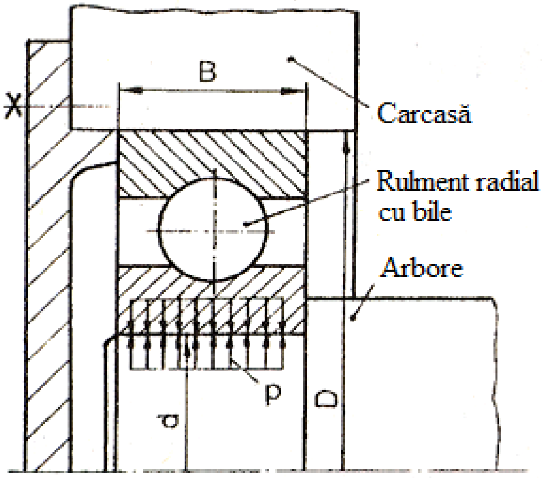
A.1. Introduction

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.

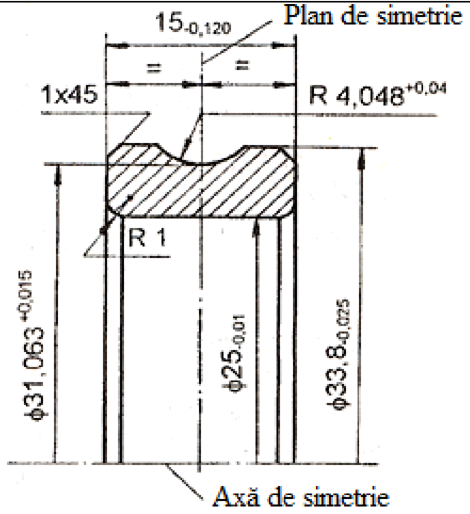
A.2. Application description

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 $E = 2.1\ 10^5$MPa, 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.

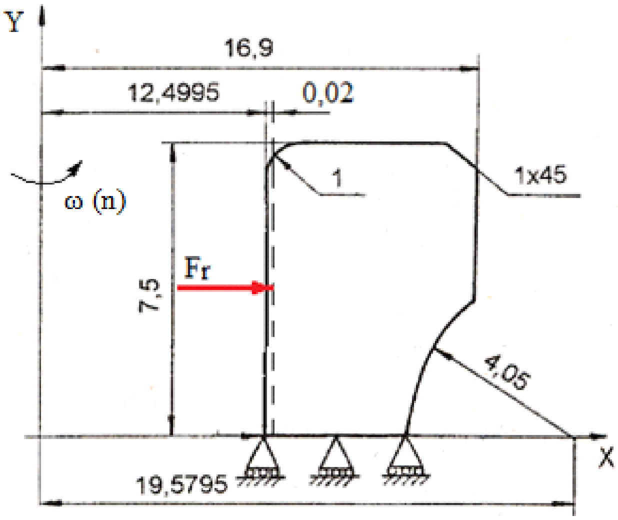
B.2. The analysis model description

The figure below shows the AEF 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 ω = πn / 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... → Save As, File name: [enter name, AEF-A.4] → ↵ 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)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)

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

C.3.2 Sketch generation

Viewing default plane (XY)

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

Rectangular lines generation

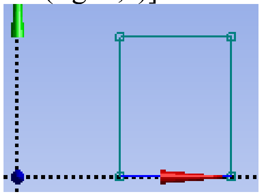
↵Draw → ↵ Rectangle → [trace rectangle line using pencil starting with, ↵ a point from left of Y axis, and finish in opposite point simultaneously with release of the mouse ↵] (fig. a)].

Generating a circle arc

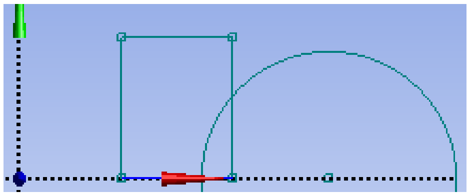
↵ Circle → [the circular line is generated by ↵ marking the center on the x-axis (coincidence symbol C appears), moving in the radial direction and releasing ↵ on the contour, fig. b].

Triming lines at the edge

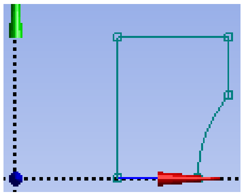
↵Modify → ↵ Trim → [activate - desactivate with ↵ the option Ignore Axis (☑/☐]) → [marking with ↵ the part to be cut (fig. b,c)]



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)].

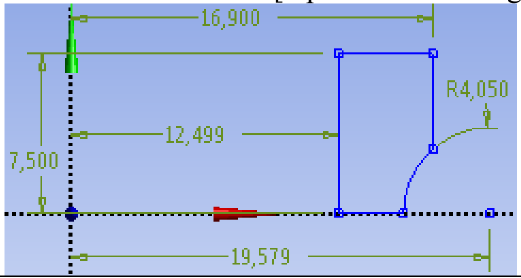
↵ Vertical → [selecting with ↵ two entities (points, lines, axes) with ↵ and the dimensions appear automatically (fig. a)] → Details View, **Dimensions: 1**: ☐ V → [introducing the dimension (fig.a)]. ↵ Radius → [selecting with the circle arc and the dimensions appear automatically (fig. a)] → Details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

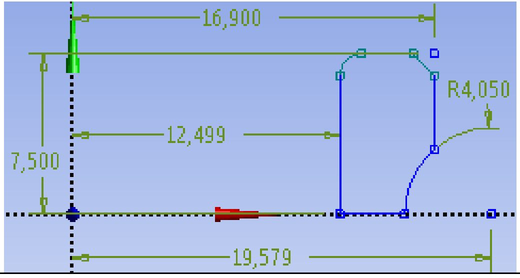
Fillet generation

↵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)].

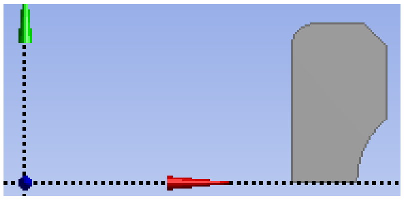




a.	*b.*

C.3.3 Surface generation

DM: ↵ 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, *Suprafaţă radială*].



C.3.4 Save of geometric model

DM: ↵ (Save Project) → ↵ X (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 → ↵ 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 → ↵ …✓ Suprafaţă bară → Details of "Suprafaţă 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).

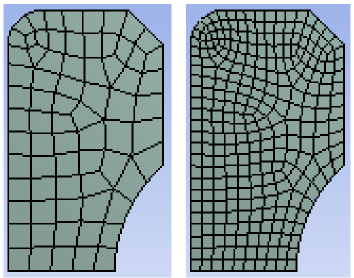
C.4.2 Model meshing

Automaticaly meshing (with default parameters)

M, 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)].



a. *b.*

Automaticaly re-meshing

└…✓ Mesh → Generate Mesh.

Visualisation of meshing statistics

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

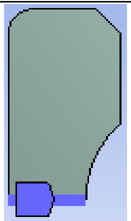
C.4.3 Supports and restraints modelling

Generating the fixed support

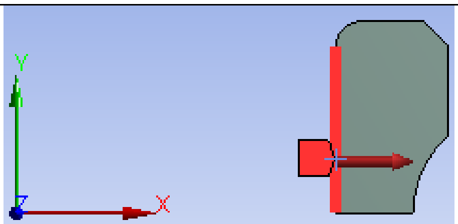
M, Outline: ↵ ⊞…? **Static Structural (A5)** → ↵ Supports ▾ → ↵ Frictionless Support; → ↵ (activating line selection filter) → [selecting with ↵ the line from OX axes (fig. a)];

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

M, Outline: ↲ ⊞···? **Static Structural (A5)** → ↲ Inertial ▾ → ↲ Rotational Velocity → Details of "Rotational Velocity": ↲ ⊟ **Scope** : ↲ Geometry → ↲ (activating face selection filter) → [selecting with ↲ suprafața/ 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

M: ↲ File → Save Project... .

D. SOLVING THE AEF MODEL

D.1 Setting the results

Selecting the directional displacements on X axis

M, 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

M: ↲ ⊞···? **Solution (A6)** → ↲ ⚡ Solve.

E. POST-PROCESSING OF RESULTS

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

Sectional / spatial view setting

↲ Λ → ↲ Tools → ↲ Options... → Λ Options: ↲ ··· Appearance, [activate – desactivate from the list ↲ ▾, ↲ Beta Options].

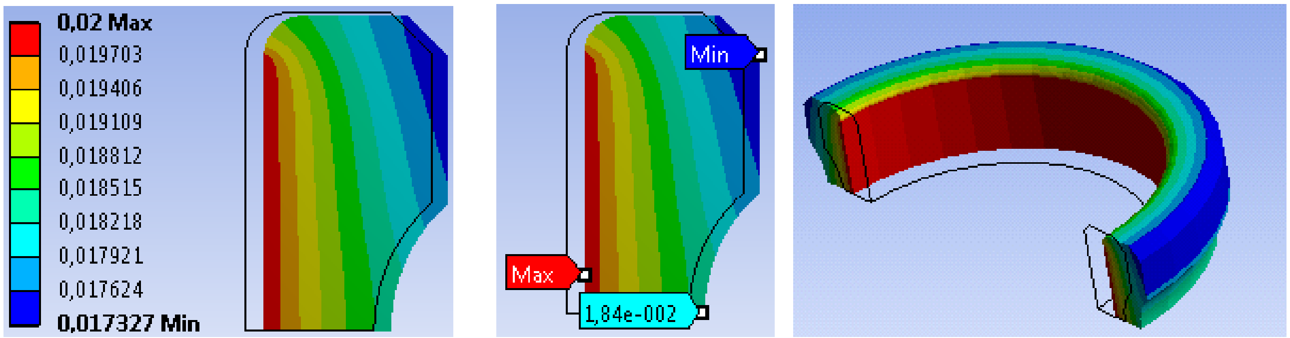
Setting the number of angular sectors

↵M → ↵ 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

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

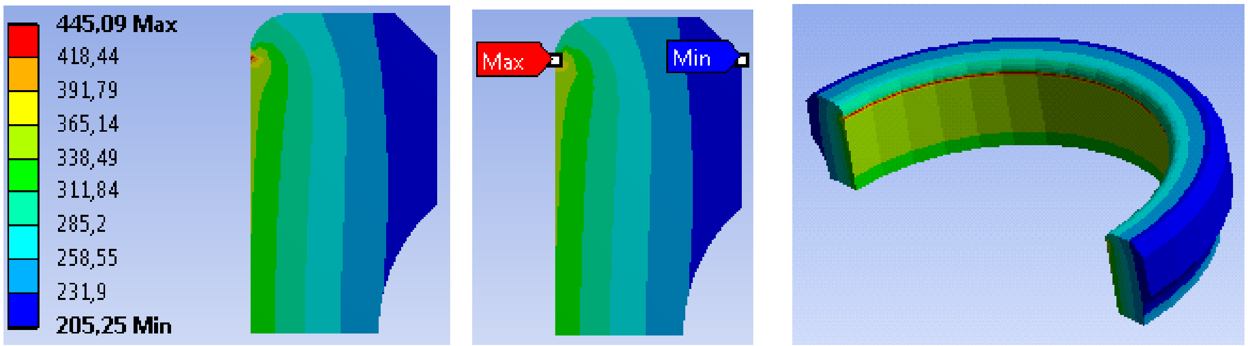
↵ Probe → [marking with ↵ the point where the value visualization is desired, 0,0184] (marking the value with label, fig. b); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. b. c.

E.3. *Viewing the equivalent stress field*

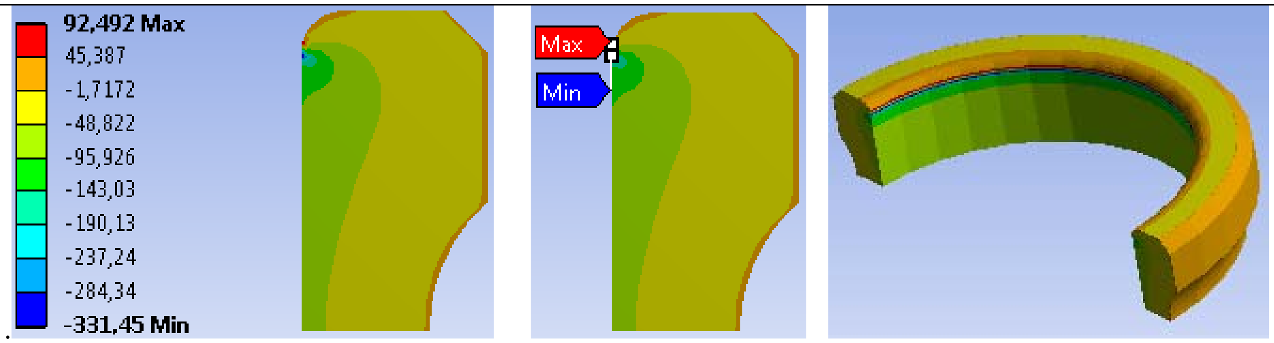
M, Outline: ↵⊞ Solution (A6) → ↵ Equivalent Stress; ↵▼ → ↵ Show Undeformed WireFrame (undeformed model view, fig. a); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. b. c.

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

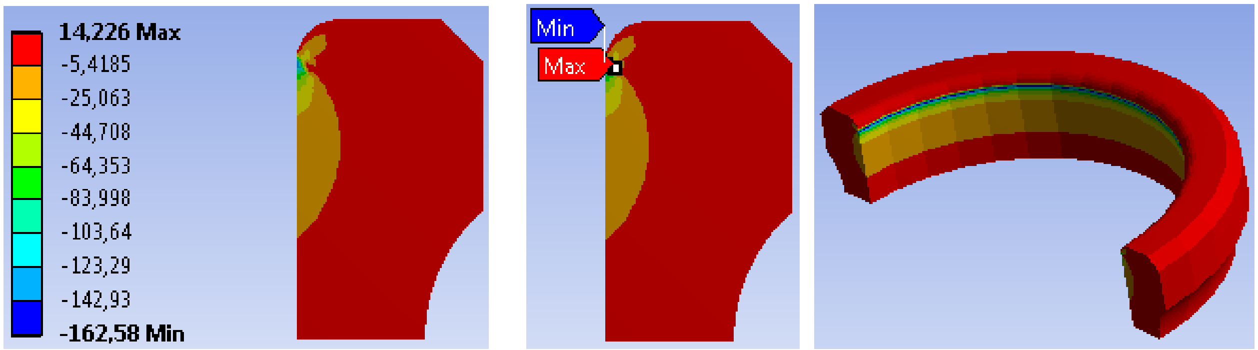
M, Outline: ↵⊞ Solution (A6) → ↵ Normal Stress; ↵▼ → ↵ Show Undeformed WireFrame (undeformed model view, fig. a); ↵ MAX (marking the node with the maximum displacement); ↵ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. *b.* *c.*

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

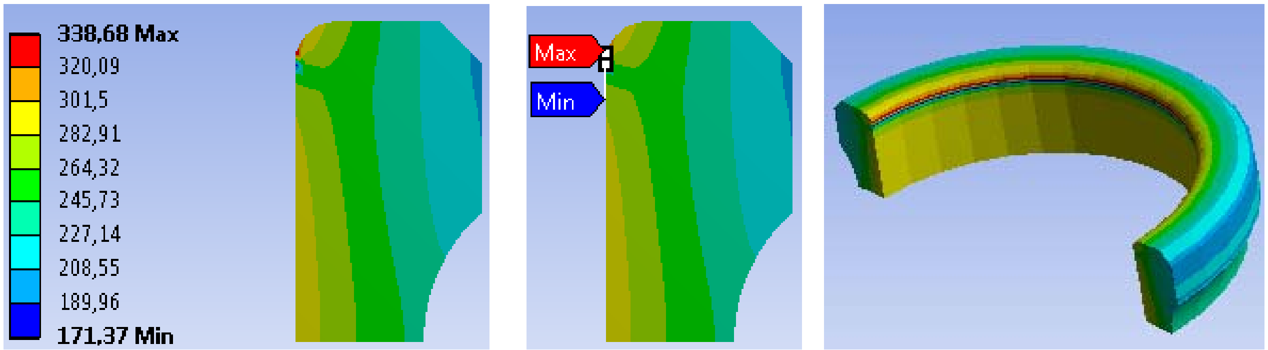
M, Outline: ↲⊞⋯? Solution (A6) → ↲⋯✓ Normal Stress 2; ↲ ▾ → ↲Show Undeformed WireFrame (undeformed model view, fig. a); ↲ MAX (marking the node with the maximum displacement); ↲ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).



a. *b.* *c.*

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

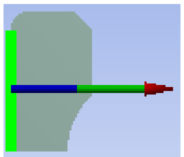
M, Outline: ↲⊞⋯? Solution (A6) → ↲⋯✓ Normal Stress 2; ↲ ▾ → ↲Show Undeformed WireFrame (view undeformed model, fig. a); ↲ MAX (marking the node with the maximum displacement); ↲ MIN (marking the node with the minimum displacement); ISO (axonometric view, fig. c).

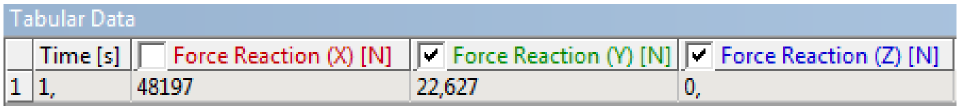


a. *b.* *c.*

E.7. Viewing the reaction force

M, 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