Application: AEF-A.7
Assembly on square profile

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

CONTENT

A. PROBLEM DESCRIPTION

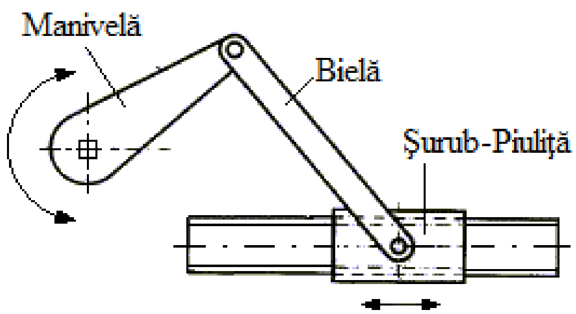
A.1. Introduction

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

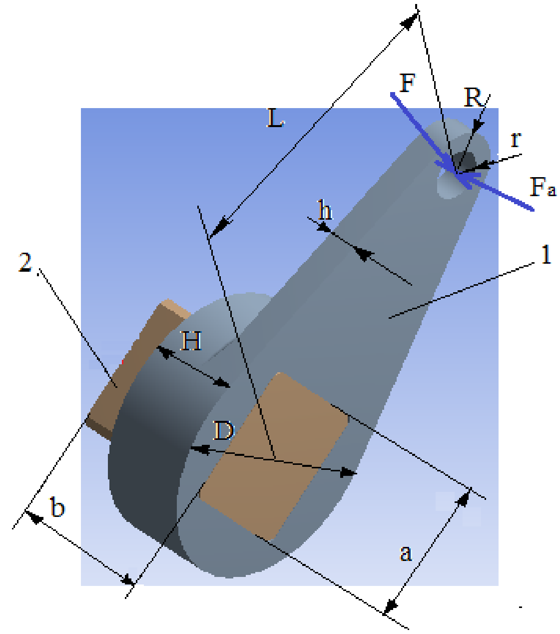
The profiled (polygonal) assemblies with the advantages of the increased load-carrying capacity and the good centering have disadvantages related to the very complex stress state in the contact areas, which requires modeling and FEA for design.

A.2. Application description

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



a.



b.

A.3. The application goal
For this application, FEA is required for the displacement and stress fields in crank 1, including the shaft assembly with shaft 2. The crank is made of E335 soft steel and the improved C45 hard steel shaft. The dimensions of the assembly elements are: L = 100, h = 10 mm, H = 30 mm, a = 45 mm, b = 50 mm, R = 16 mm, r = 8 mm. The crank is loaded with tangential force F = 10000 N and axial F_a = 2000 N.

B. THE FEA MODEL

B.1. The model definition

In order to design the crank FEA model, it is necessary to consider and a portion of the driven shaft adopting the following simplifying hypotheses:

- considering friction in mechanical contacts,
- adoption of material strength constraints (embedding, concentrated force action),
- the material has elastic linear behavior,
- the deformation occurs static (the variation of the deformation force with time is not taken into account).

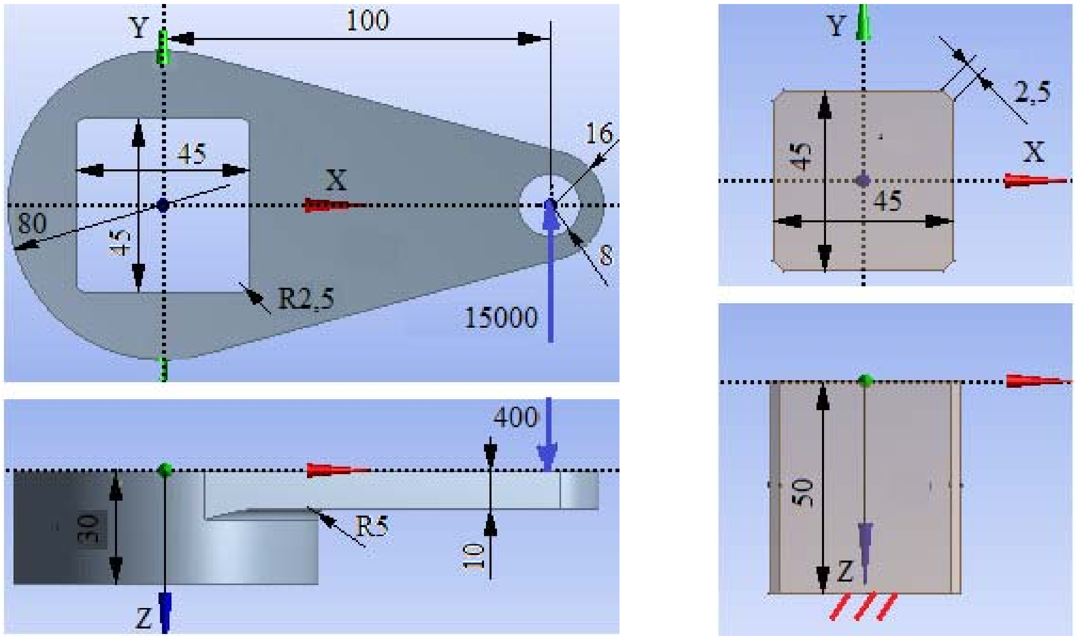
B.2. The analysis model description

The model for analysis is based on the 3D geometric model of the crank (fig. a) in contact with friction with the 3D model of the shaft (fig. b). For analysis the structure is composed of two solids (crank and shaft) which are generated by extrusion and discretized with 3D finite elements.

In order to make the *mechanical contacts between the crank and the shaft* on flat surfaces, the square profile of the crank will be connected to the corners with 2.5 mm radius and the shaft one will be 2.5 mm (fig. A, b).

In order to simulate the behavior of the assembly as close to reality when modeling the mechanical contacts between elastic deformable surfaces, the friction coefficient μ = 0.2 will be considered.

The loading of the model for the analysis will be carried out in the area of the crankshaft coupling (subchapter A.2, fig. a) with the tangential forces, 20000 N, and axial 4000 N (fig. a, b). The modeling of the loads with these forces will be done with the specific function "Bearing Load".



a. ***b.***

B.3. Characteristics of the material and the environment
For the analysis with finite elements the strength characteristics of the materials are: • longitudinal modulus of elasticity E = 205000 N / mm^2 (MPa), coefficient of transverse contraction (Poisson) ν = 0.29, for steel E335 (σ_{02} = 335MPa – at traction; σ_{02} = 400 MPa – at compresion; σ_r = 590…760 MPa) associated with Manivelă element. • the longitudinal elasticity modulus E = 210000 N / mm2, the coefficient of transverse contraction (Poisson) ν = 0.3, for the C45 alloy carbon steel (0.4% C) associated with the Arbore solid which, after the hardening treatment, reaches at hardness 250 ... 280 HB (σ_{02} = 520MPa – at traction; σ_{02} = 560 MPa – at compresion; σ_r = 690…860 MPa) The average working temperature of the subassembly, $T_0 = 22^0$ C.

C. PREPROCESSING OF FEA MODEL

C.1 Activarea şi salvarea proiectului / Creating, setting and saving the project
Creating of the project , Toolbox: ↵⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [the name can be changed Static Structural in *Assem_sq*]. *Problem type setting (2D)* A : ↳ Geometry → ↵Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵Analysis Type, [select from list with ↵, ↵ 3D] (this setting is usually default) → [close the window ↵ X]. *Save of the project* ↵ Save As... → Save As, File name: [input name, AEF-A.7] → ↵ Save.

C.2 Modelling of material and environment characteristics
Generating of the material characteristics for the component Manivelă , Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Click here to add a new material → [input name: E335] (it apear in line E335) → ↵ E335 → Toolbox: ↵ ⊞ Linear Elastic → ↵↵ Isotropic Elasticity → Table of Properties Row 2: Isotropic Elasticity: Temperature (C) → select from list with ↵ C (grade Celsius) input value, 20], Young's Modulus (Pa) → [select from list with ↵

MPa, input value 205000], Poisson's Ratio → [input value 0,29] (the window is automatically generated Properties of Outline Row 4: E335 and the graph Chart of Properties Row 2: Isotropic Elasticity, in which the data entered are highlighted).

Generating of limit and permissible characteristics of the material E335

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

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

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

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

Generating of the material characteristics for the component Arbore

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

Generating of limit and permissible characteristics of the material C45

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

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

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

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

Update of the database and exit

: ↵ Update Project → ↵ Return to Project.

C.3. Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

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

C.3.2 Sketch 1 generation (Shaft)

Viewing default plane (XY)

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

Generating of rectangular contour

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

Centering of rectangular contour

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

Dimensioning of rectangular contour

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

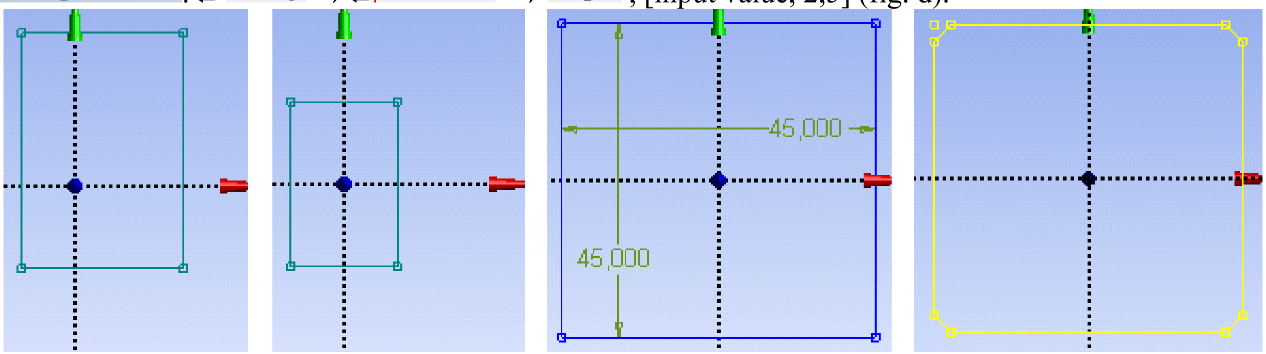
↲ Vertical → [select with ↲ liniile paralele cu X / paralle lines with X] (the dimension is automatically viewed) → Details View, Dimensions: 2 : ↲ V → , [input value, 45] (fig. c);

Changing the dimension view

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

Generating the chamfers

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



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

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

Generating sketch 2

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

Generating overlapping lines over Sketch1

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

Generating the chamfers

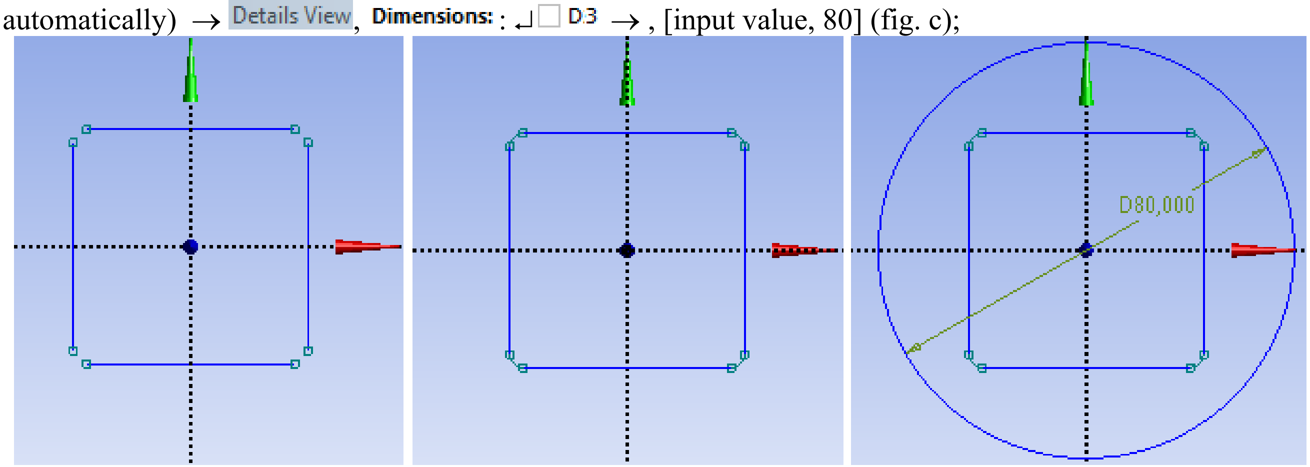
Sketching Toolboxes: ↲ Modify → ↲ Fillet → Radius: , [input value, 2,5] → [two adjacent lines are selected with ↲ at a time and the connections automatically appear] (fig. b).

Generating the circular line

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

Dimensioning of the circle

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



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

C.3.4 Generate sketch 3 (arm)

Setting the Sketch 3

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

Generating overlapping circular lines over Sketch2

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

Generating circular lines

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

Generating straight lines (tangents)

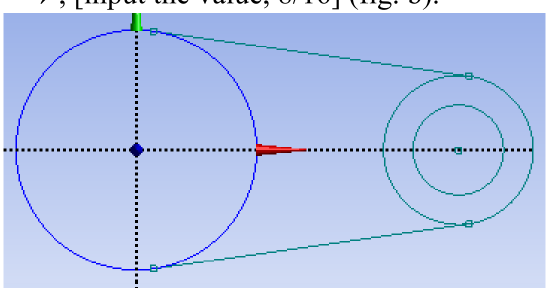
Sketching Toolboxes: ↵ Draw → ↵ Line by 2 Tangents → [the two circles are selected with ↵ in a row and the tangent lines automatically appear] (fig. b).

Generating contour

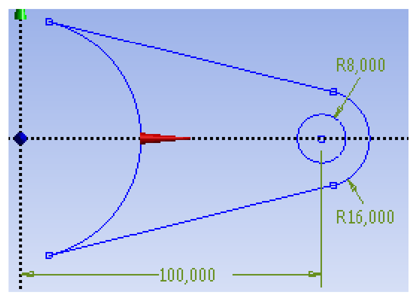
Sketching Toolboxes: ↵ Modify → ↵ Trim → [the parts that do not belong to the contour are deleted] (fig. a, b).

Dimensioning the new entities

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



a.



b.

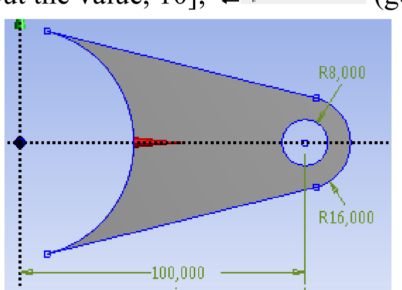
C.3.5 Generating of the surface and body of the crank arm

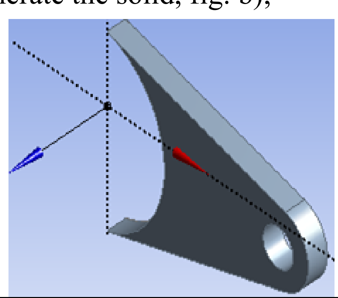
Generating of the surface for the arm

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

Generating of the solid body of the arm

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





a.	b.

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

Generating the surface for the hub

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

Generating of the solid body of the hub

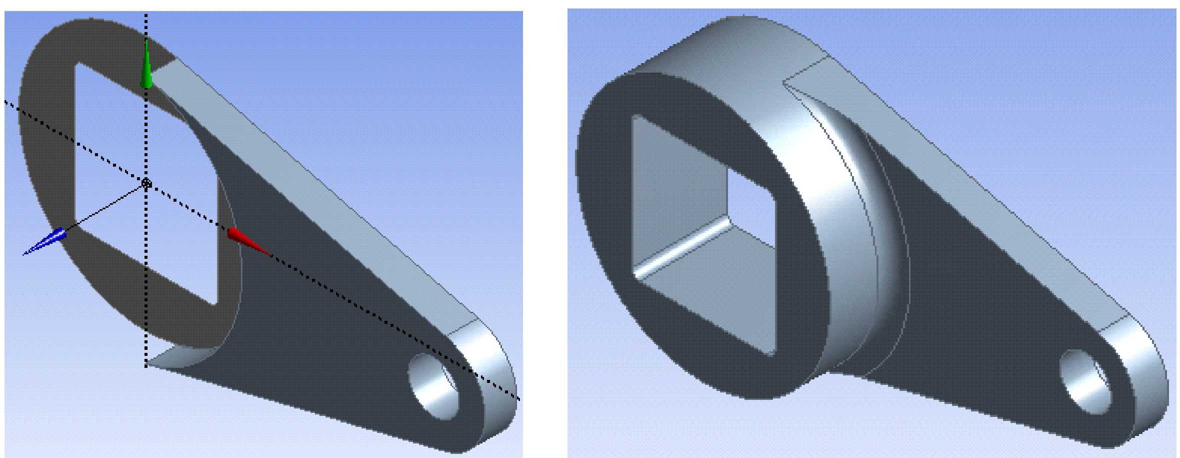
↵ Extrude → ↵ Surface Body (the surface associated with the hub)→ Details View, Details of Extrude: ↵↵ Geometry → ↵ Apply; FD1, Depth (>0) → [input the value, 30]; ↵ Generate (generate the solid, fig. b);

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

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

Generating the radius connection

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



a. b.

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

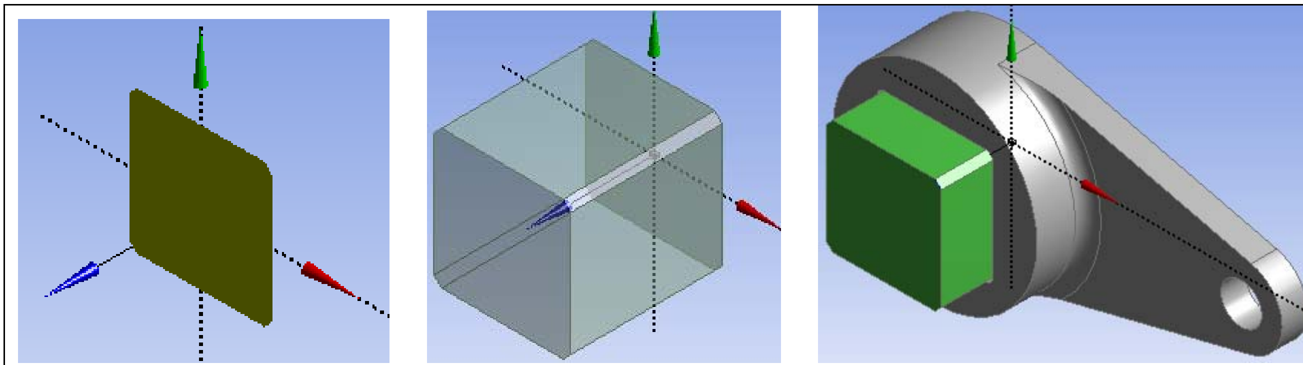
Generating the surface for the shaft

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

Generating the solid body of the shaft

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

↵ Surface Body → ↵ Suppress Body (supress body).



a. *b.* *c.*

C.3.8 Save of geometric model

DM: ↵ Save Project) → ↵ (Close).

C.4. Finite element modelling

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

Launching of the finite element modeling module

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

Disabling redundant entities

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

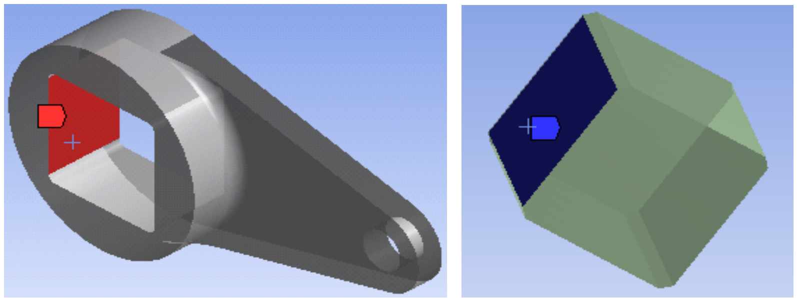
Obs. In the specification tree, we observe, as a consequence of the connections between the two bodies, that a connection Contact Region has been automatically generated in the subdivision Connections, which will be further personalized.

C.4.2 Modelling the friction connections

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

For each of the other 3 contacts, the sequence will be followed: ↳ Contacts → ↵ Insert → ↵ Manual Contact Region → ... (continue according to the sequence above, but for another pair of surfaces:,). After browsing the above sequences in the specification tree Contacts it appears Frictional - Manivela To Arbore for four times.

Obs. For a good convergence of the solution, the entities (surfaces or edges) belonging to the fixed bodies, to the bodies with increased material rigidity (greater longitudinal elasticity modulus) or smaller curves are adopted in the window Details of "Frictional at the option.



a. *b.*

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

Setting the local discretization parameters in the contact areas

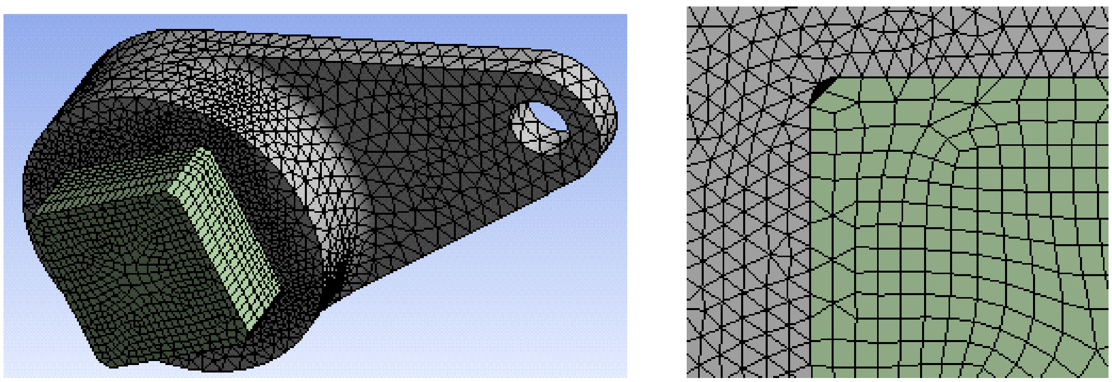
M, Outline: ↵ Mesh → Details of "Mesh", ↵⊞ **Defaults**: ↵☐ Relevance → [will be adopted with ↵ ▭ the value, 100]; ↵⊞ **Sizing**: ↵Relevance Center → [select from list ↵▾, ↵Fine].

Setting the local parameters in the contact areas

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

Automatic discretization

↳ Mesh → ↵ Generate Mesh (fig. a, b).

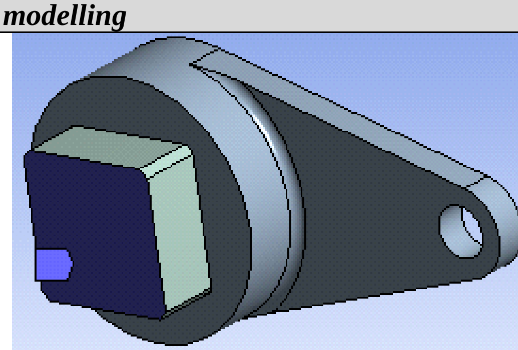


a. *b.*

C.5. *Supports and restraints modelling*

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

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



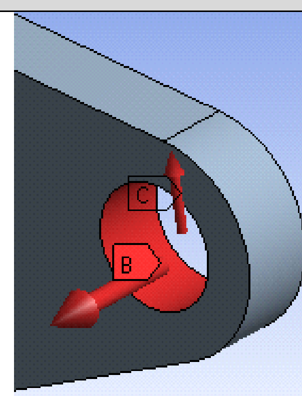
C.6. *Loads modelling*

Generating of force load on the surface

M, Outline: ↳ **Static Structural (A5)** → ↵ Insert → ↵ Force → Details of "Force", **Scope**: ↵Geometry → ↵ → [the cylindrical surface of the bore in the crank arm shall be selected with ↵] → ↵ Apply; **Definition**: ↵Define By → [select from list with ↵▾, ↵Components]; ↵Z Component → [input the value, 400] (force B).

Generating of load with force in the bearing

↳ **Static Structural (A5)** → ↵ Insert → ↵ Bearing Load → Details of "Bearing Load", **Scope**: ↵↵Geometry → ↵ →



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

D. SOLVING THE FEA MODEL

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

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

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

D.2 Setting the results

Setting the total displacement

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

Setting the equivalent stress

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

Setting the structural error

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

Setting parameters from contacts

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

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

Setting the design parameters

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

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

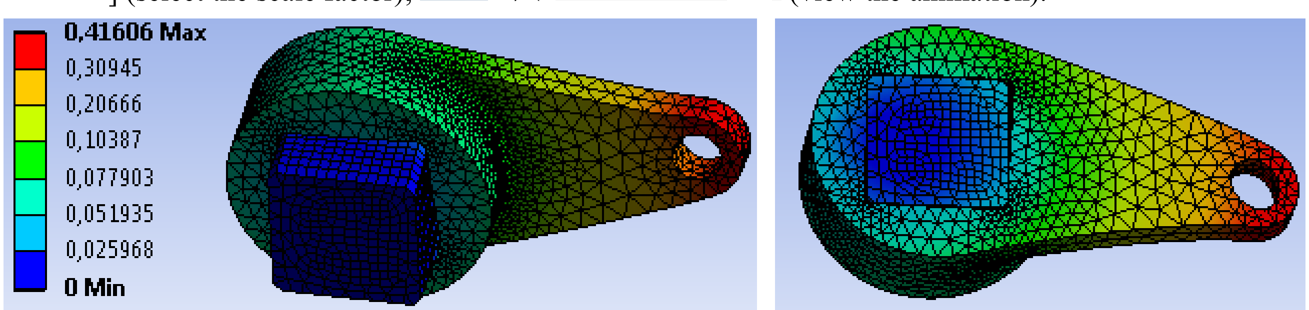
D.3 Launching the solving module

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

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement field

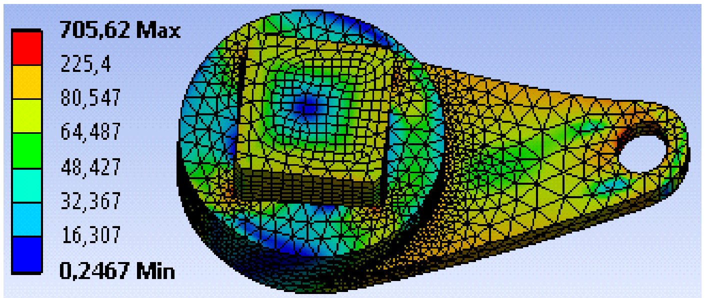
, Outline: ↵⊞ Solution (A6) → ↵ Total Deformation (fig. a); ↵ ISO (axonometric visualization); ↵ → [select from list with ↵▾, ↵ Smooth Contours] (visualization of smooth contours); ↵ → [select from list with ↵▾, ↵ Show Elements] (visualisation the FE structure). ↵Result → [select from list with ↵▾, ↵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

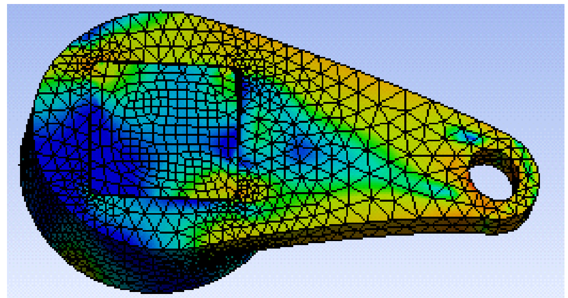


E.2 Viewing the equivalent stress field

Viewing the equivalent stress field

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





a. ***b.***

Viewing the equivalent stress field in section

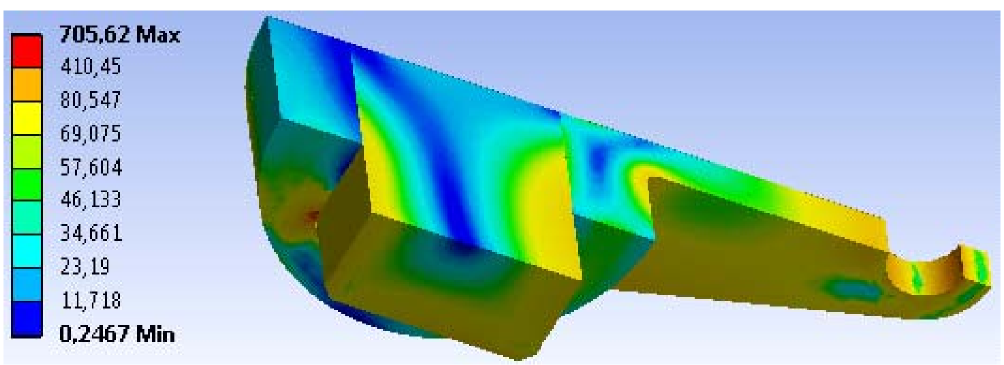
Generating of section plan

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

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

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

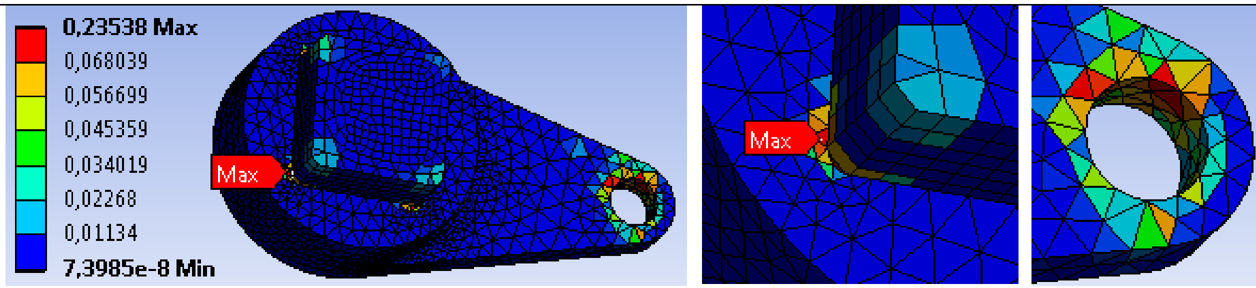
Obs. In order to delete the section plan, the succession is carried out: ↵ → Section Planes: ↵ Slice Plane 1 → ↵ ✕ → ↵ .



c.

E.3 Viewing the structural error

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



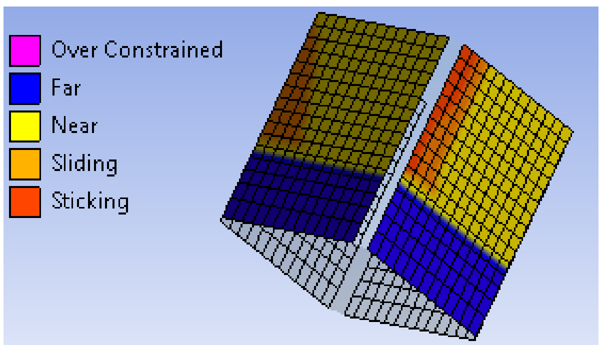
E.4 Viewing the fields of the contact parameters (state, friction stress, pressure, sliding, jump, penetration)

Solution (A6) → Status (fig. a);

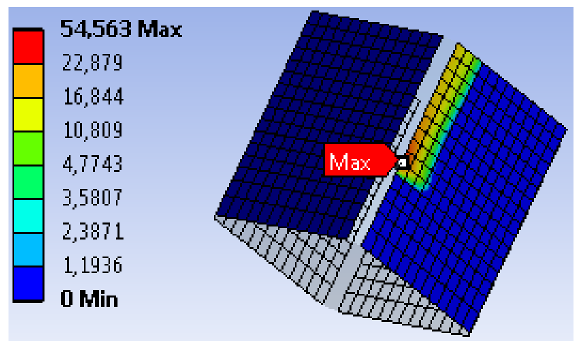
Frictional Stress (fig. b); Pressure (fig. c);

Sliding Distance (fig. d);

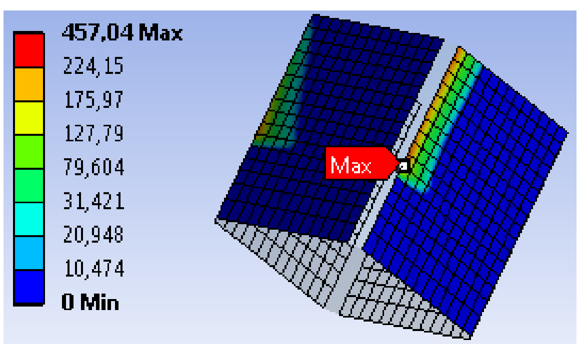
Gap (fig. e); Penetration (fig. f).



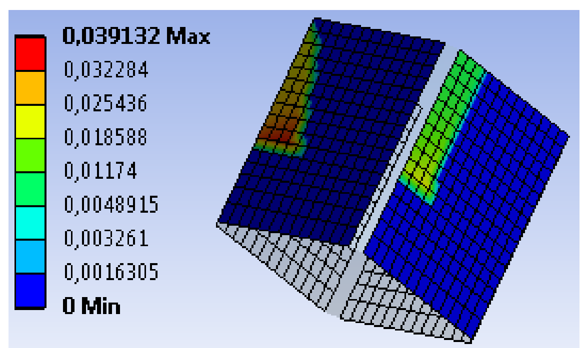
a.



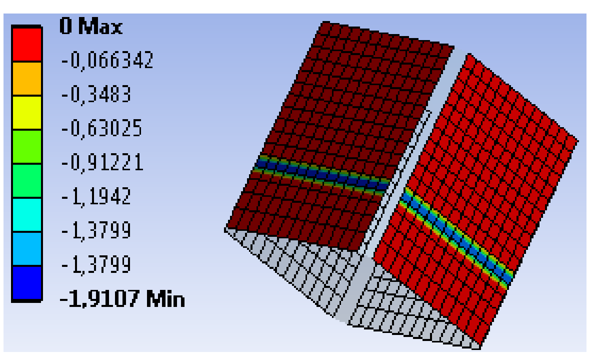
b.



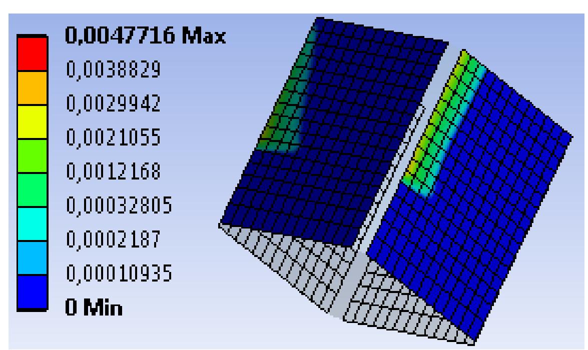
c.



d.



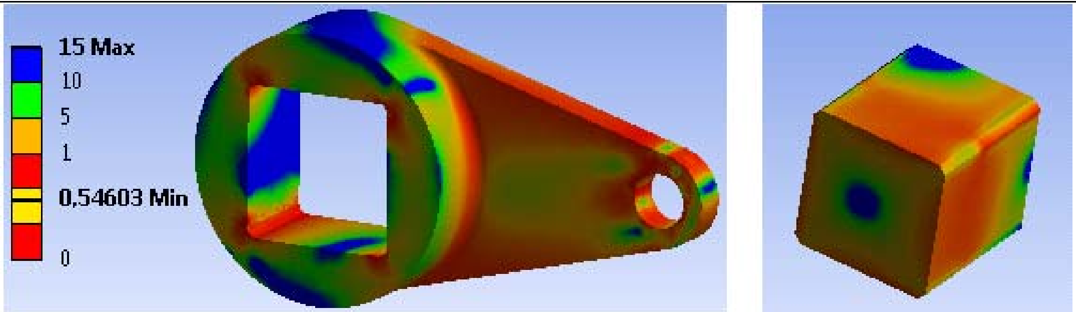
e.



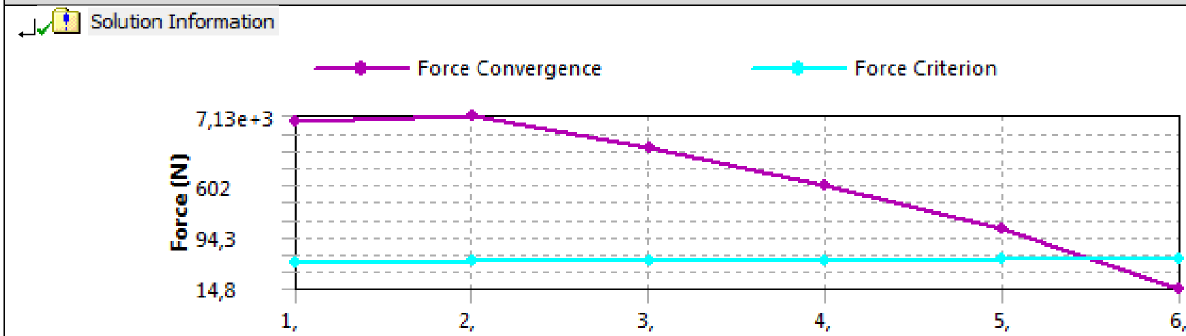
f.

E.5 Viewing the safety factor field

Solution (A6) → Safety Factor;



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



F. ANALYSIS OF RESULTS

F.1 Interpretation of results

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

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

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

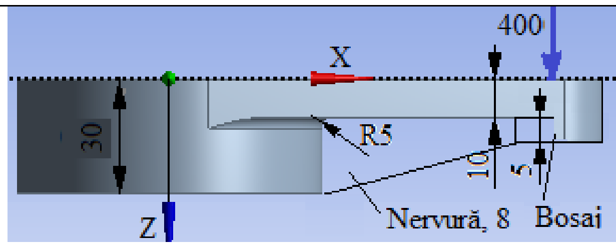
Following the analysis of the obtained results, related to precision and convergence, as a result, of the modeling and AEF (subcap. E.3 and E.6) the following are highlighted:

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

F.3 Design studies

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

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



G. CONCLUSIONS

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

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

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