Application: AEF-A.6
Plastic deformation

KEY WORDS
Nonlinear static analysis, Spatial state of stresses, Nonlinear material, 3D geometric model, 3D finite element, Nonlinear finite element (parabolic), Cylindrical coordinate system, Mechanical contact without friction, Structural error, Plastic deformation Subset of processing

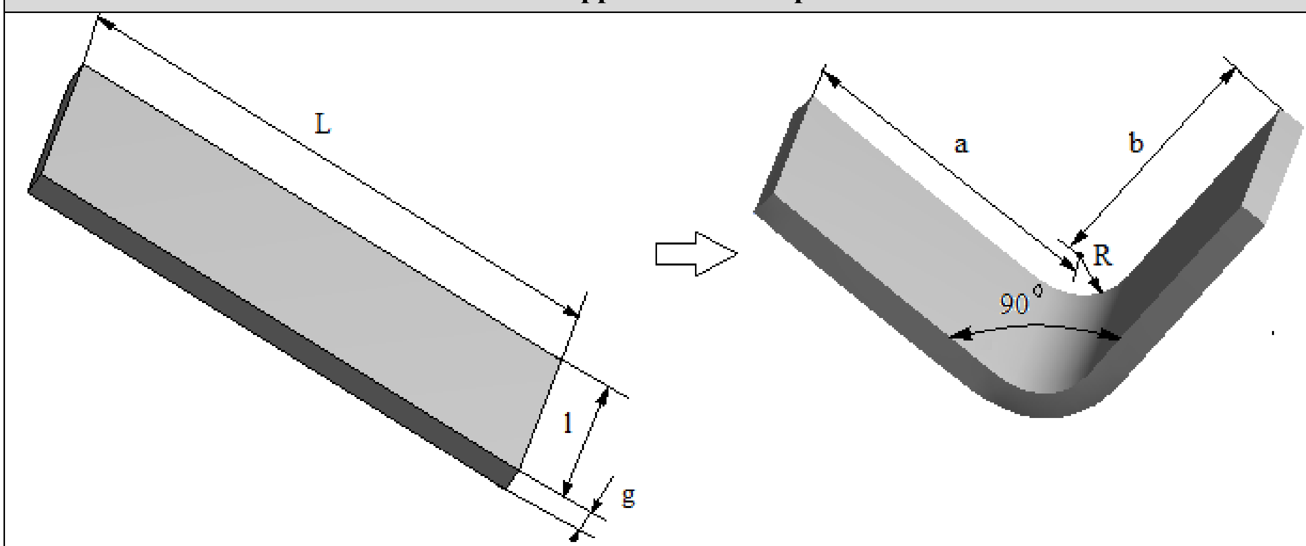
CONTENT

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 *cold plastic deformation processes* of the thick sheets that assume the material parameters that describe the nonlinear *behavior with remaining deformations.*

A.2. Application description



The *cold bending* of the flatbed (the blank) in order to obtain the 90 ° corner piece with unequal wings implies the use of a die-punch device which involves fixing one wing and the plastic deformation of the other wing by means of the punch pressing it on the fixed die. After removing the punch, the piece remains in a deformed state. The material of the band is a soft (ductile) steel that involves *increased plastic deformation capacity* in interaction with the active parts of the device which are made of hardened non-plastic steel.

A.3. The application goal
This application assumes the FEA of the *bending process of a flat panel* with the length L = 105.7 mm, the width l = 40 mm and the thickness g = 5 mm in order to obtain a corner at 90o with uneven wings a = 40 mm and b = 50 mm. In the case of this application, it is necessary to establish *the maximum deformation load F* without having an excessive flow or the break established by the values of the maximum stresss that appear in the critical areas. In addition, following the analysis will be followed the determination of the values of the *pressures* in the interaction zones of the semi-manufactured with the active elements (die, punch) of the deformation device, necessary for its design.

B. THE FEA MODEL

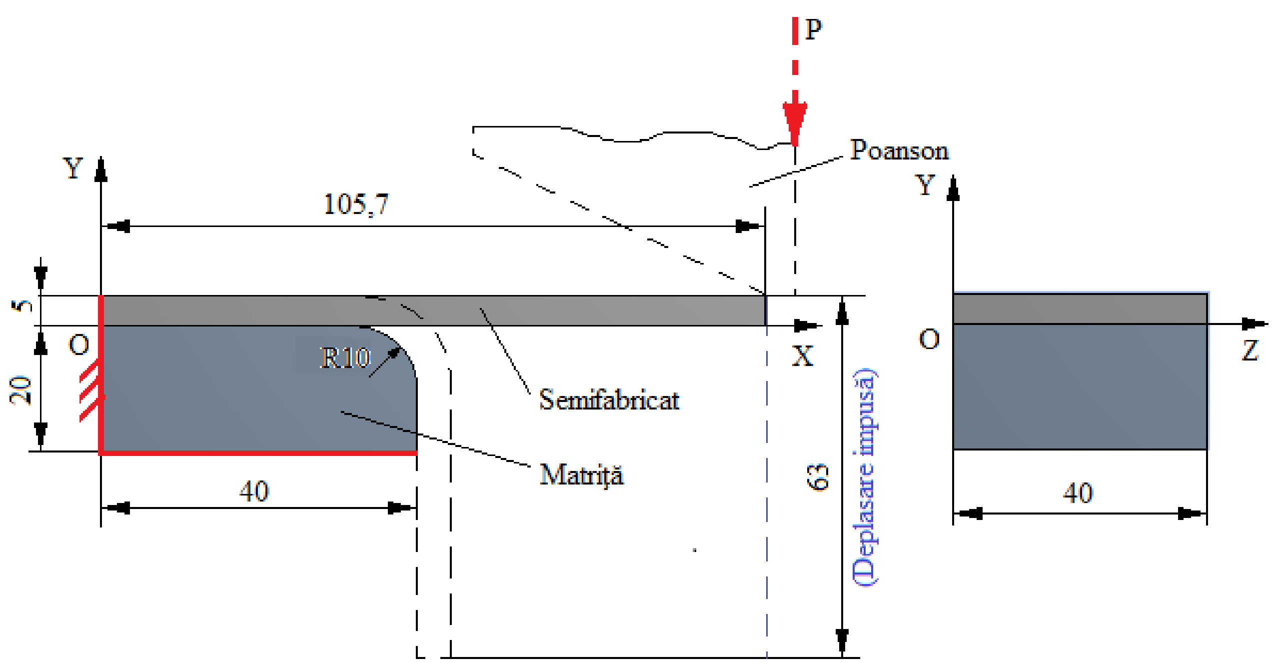
B.1. The model definition

In order to design the FEA model, it is also necessary to consider the die-punch deformation device, adopting the following simplifying hypotheses:

- neglecting the effects of friction in mechanical contacts,
- adoption of material strength constraints (embedding, concentrated force action),
- the material has nonlinear elasto-plastic behavior according to a bilinear scheme.
- the deformation takes place 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 half-finished element in contact without friction with the 3D model of the active area of the mold. For analysis, the structure is composed of two solids that are modeled with 3D finite elements.
In order to simulate the plastic deformation as close to reality as possible, it will be necessary to move the edge of the half-finished element with the value -63 mm, in the direction of the axis of action of the punch. This constraint (displacement imposed) considered as an indirect load leads after the analysis to determine the value of the pressing force of the punch P.



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 = 203000 N / mm2 (MPa), coefficient of transverse contraction

(Poisson) ν = 0.29, modulus of plasticity Ep = 1800 MPa for steel of mechanical construction E295 (σ02 = 295MPa, σr = 490 ... 660 MPa) associated with the half-finished element.

- the longitudinal elasticity modulus E = 210000 N / mm2, the coefficient of transverse contraction (Poisson) ν = 0.3, for the 40Cr10 alloy carbon steel (0.4% C and 1% Cr) associated with the die (Matrita) solid which, after the hardening treatment, reaches at hardness 50 ... 55 HRC.

The average working temperature of the subassembly, $T_0 = 22^0$ C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project

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

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].

Saving of the project

↵ Save As... → Save As, File name: [input name, AEF-A.6] → ↵ Save.

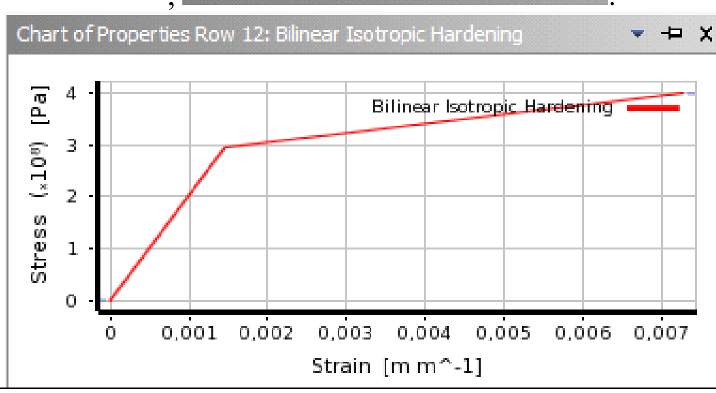
C.2 Modelling of material and environment characteristics

Generating of solid material characteristics Semifabricat

, Project Schematic: ↳ Engineering Data ✓ → ↵ Edit... → Outline of Schematic A2: Engineering Data: ↵ Structural Steel → [the name will be changed to *Semifabricat*] (the features are set by default and the values will change); Toolbox: ↵ Bilinear Isotropic Hardening ; Properties of Outline Row 3: Semifabricat: Young's Modulus → [select from list with MPa / input value: 203000], Poisson's Ratio → [input value:/ 0,29]; Bilinear Isotropic Hardening → Table of Properties Row 12: Bilinear Isotropic Hardening: Temperature (C) → [select from list with ↵ C (grade Celsius) / input value: 22], Yield Strength (MPa) → [select from list with ↵ MPa / input value 295], Tangent Modulus (Pa) → [select from list with ↵ MPa / input value 1800] (the window below is automatically generated).

Generating of solid material characteristics for the mold

Outline of Schematic A2: Engineering Data: ↵ Click here to add a new material → [the name of the *mold* is entered and the feature set appears,];, Matriță]; Toolbox:↵ Isotropic Elasticity ; Table of Properties Row 2: Isotropic Elasticity: Temperature (C) → [select from list with ↵ C (degree Celsius) / input value: 22], Young's Modulus (Pa) → [select from list with ↵ MPa / input value: 210000], Poisson's Ratio [input value: 0,3] (you can see the generation of these values as well as of others dependent on them and in the window Properties of Outline Row 4: Matriță). ↵ Update Project → ↵ Return to Project.



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 Plane);

Generating of rectangular contour Semifabricat

Sketching Toolboxes: ↵ Draw → ↵ Rectangle → [the rectangular line is generated in quadrant I, marking, with ↵, the first corner in the center of the coordinate system (coincidence symbol P appears) and the release ↵ in the opposite corner] (fig. a).

Dimensioning of outline sketch of Semifabricat

Sketching Toolboxes: ↵ Sketching → ↵ Dimensions → ↵ Semi-Automatic → [the semiautomatic dimensions are generated by marking with ↵); Display → [deactivate option Name: ☐ activate option Value: ☑] (the values of the dimensions on the drawing will be displayed); Details View, **Dimensions: 2**: ☐ L1 → [input value: 5], ☐ L2 → [input value: 40] (fig. a).

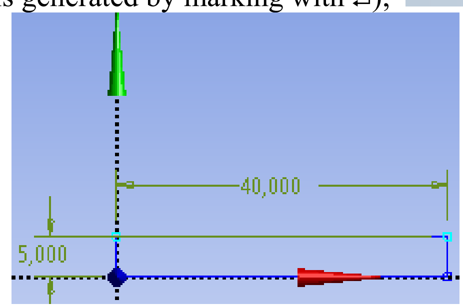
Generating of rectangular contour Matrita

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

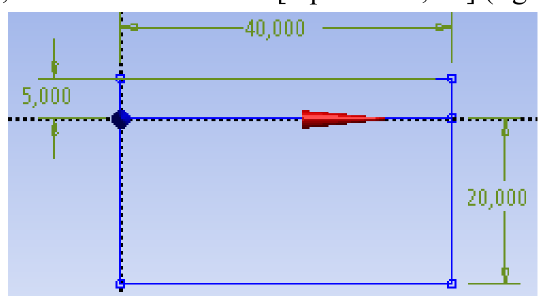
Sketching Toolboxes: ↵ Draw → ↵ Rectangle → [a rectangular line is generated with the side common to the previous sketch (coincident with the OX axis) by pressing ↵ the first corner in the lower left corner of the previous rectangle (coincidence symbol P appears and on overlapping with the axis OX symbol C) and releasing ↵ in the opposite corner] (fig. b).

Dimensioning the rectangular sketch of Matrita

↵ Sketching → Sketching Toolboxes: ↵ Dimensions → ↵ Semi-Automatic → [the dimension associated with the thickness is generated by marking with ↵); Details View, **Dimensions: 1**: ☐ L1 → [input value, 20] (fig. b).



a.



b.

C.3.3 Generating the solids objects (Semifabricat, Matriţă)

Generating of solid Semifabricat

: ↵ Modeling → ↵ Sketch1 ; ↵ (isometric view); ↵ (masking sketches);

↵ Extrude → Details View: **Details of Extrude1**, Geometry → ↵ Apply; ☐ FD1, Depth (>0) → [input value 105,7];

↵ Generate (fig. a).

↵ Solid → Details View, **Details of Body**, Body → [the default Solid name is changed to Semifabricat].

Generating of solid Matriţă

↵ Modeling → ↵ Sketch2 ;

↵ Extrude → Details View: **Details of Extrude2**, Geometry → ↵ Apply; ☐ FD1, Depth (>0) → [input value 50];

Operation → [select from list with ↵ Add Frozen] (solid separated from the previous one will be generated) ;

↵ Generate (fig. b).

↵ Solid → Details View, **Details of Body**, Body → [modify name: Solid în to Matriţă].

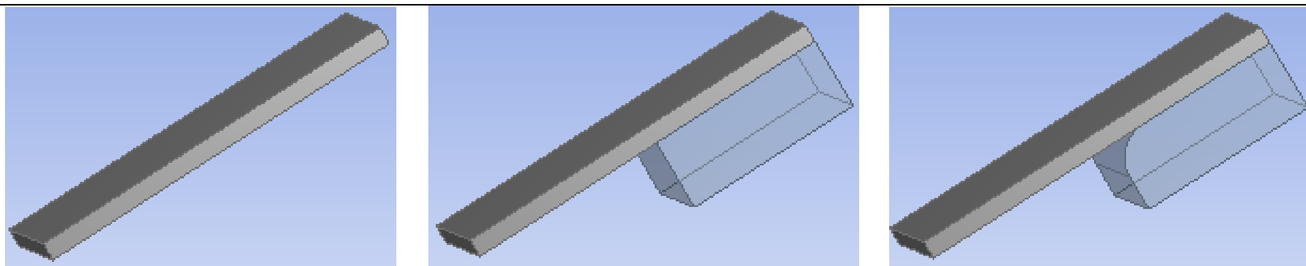
Generating of solid radius

↳ Semifabricat → ↵ Hide Body (hide the solid Semifabricat) ; Tree Outline: ↵ Matriţă → ↵ → [it will be selected with ↵ the edge that will rotate] ;

↵ Blend → ↵ Fixed Radius; Details View, **Details of FBlend1**, ☐ FD1, Radius (>0) → [input value 10];

Geometry → ↵ Apply; ↵ Generate (fig. c).

↳ Semifabricat → ↵ Show Body (view solid Semifabricat).



a. *b.* *c.*

C.3.4 Saving of geometric model

DM: ↵ (Save Project) → ↵ (Close Design Modeler).

C.4. Finite element modelling

C.4.1 Launching of the finite element modeling module

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 → ↵ Semifabricat → Details of "Semifabricat", **Material** : ↵ Assignment → [select from list with ↵, ↵ Semifabricat] .

↵ Matriţă → ↵ Details of "Matriţă", **Material** , ↵ Assignment → [select from list with ↵, ↵ Matriţă] .

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

C.4.2 Modeling the contact type

Generating of contact Semifabricat-Matriţă

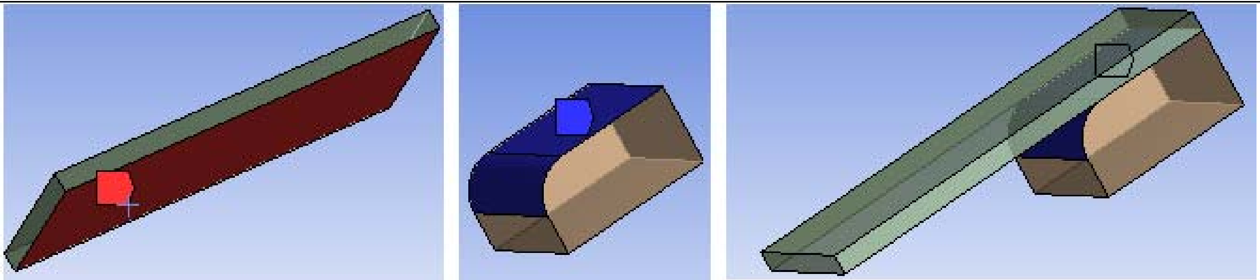
, Outline: ↵ Connections → ↵ Contact Region → Details of "Contact Region", **Definition**: ↵ Type → [select from list with ↵, ↵ Frictionless].

Obs. If the initial contact generation command does not appear automatically, to initiate the contact Contact Region, the sequence is followed: ↵ Connections → ↳ Contacts → ↵ Insert → ↵ Manual Contact Region, after which it is customized as above.

↵ Frictionless - Semifabricat To Matriţă → ↳ Matriţă → ↵ Hide Body (hide the solid Matriţă) → ↵ →[is selected with ↵ the lower face of the entity Semifabricat, fig.a]→ Details of "Frictionless - Semifabricat To Matriţă", **Scope**: ↵ Contact → ↵ Apply (option Contact Bodies indexing automatically, Semifabricat);

↳ Matriţă → ↵ Show Body → ↳ Semifabricat → ↵ Hide Body (hide the solid Semifabricat) → ↵ → [select with ↵+Ctrl the initial contact seating face and the connecting surface, fig. b] → Details of "Frictionless - Semifabricat To Matriţă", **Scope**: ↵ Target → ↵ Apply (option Target Bodies it is indexed automatically, Matriţă); ↵ **Definition**: ↵ Behavior → [select with ↵, ↵ Symmetric]; ↵ **Advanced** → ↵ Formulation → [select with ↵, ↵ Augmented Lagrange] (method of solving the nonlinear model).

Obs. For a good convergence of the solution is adopted in the window at the Details of "Frictionless - option Target, in accordance with the Target entities (surfaces or edges) belonging to the fixed bodies, to the bodies with increased material rigidity (the longitudinal elasticity module may large) or have smaller curves.



a. *b.* *c.*

C4.3 Setting discretization parameters, model discretization and analysis type setting

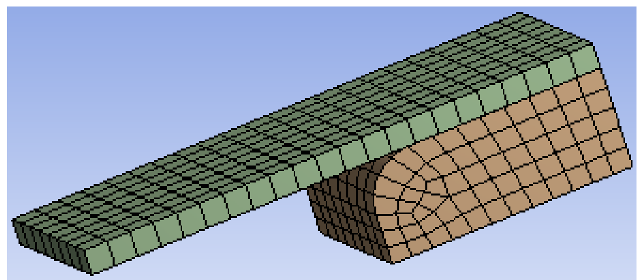
Setting the local discretization parameters in the contact areas

Outline: ↳ Mesh → ↵ Insert → ↵ Sizing ; ↳ Matriță → ↵ Hide Body → ↵ → [select with ↵ the lower face of the entity Semifabricat]; ↵ Sizing → Details of "Sizing" - Sizing Scope: ↵ Geometry → ↵ Apply ; Definition → ↵ Element Size → ↵Default, [input value, 5].

↳ Mesh → ↵ Insert → ↵ Sizing ; ↳ Matriță → ↵ Show Body → ↳ Semifabricat → ↵ Hide Body → ↵ → [select with ↵+Ctrl the initial contact seating face and the connecting surface]; ↵ Sizing → Details of "Sizing" - Sizing: ⊟ Scope , ↵ Geometry , ↵ Apply ; Definition: ↵ Element Size ↵Default, [input value, 5].

Automatic meshing

↳ Mesh → ↵ Generate Mesh.



Setting the analysis parameters

↵ Analysis Settings → Details of "Analysis Settings",

↵⊞ Step Controls: ↵Number Of Steps → [input value, 7];

↵⊟ Solver Controls : Large Deflection, → [select with ↵ , ↵On].

Obs. The displacements have large values and geometric type nonlinearity is adopted Large Deflection.

C.5 *Supports and restraints modelling*

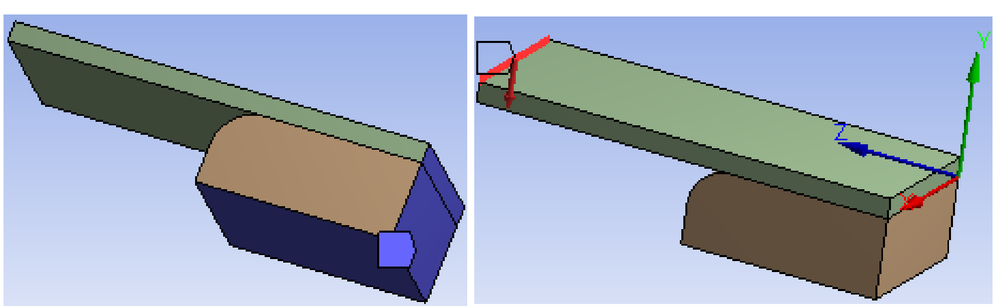
Generating of the constraint type (cancels all 6 degrees of mobility)

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

Generating of forced displacement constraint

↵⊞ Static Structural (A5) → ↵ Supports ▾ → ↵ Displacement ; ↵⊞ Model (A4) → ↵ → [the edge of the entity Semifabricat on which the *Poanson* is pressed] ; ↵ Displacement → Details of "Displacement", Scope: ↵Geometry → ↵ No Selection → ↵ Apply ; ↵ Y Component → ↵ →

[select from list ↵ , ↵Tabular] → Tabular Data → [input value in column ☑ Y [mm] valorile 0, -9, -18, … -63] (fig. c).



Tabular Data

	Steps	Time [s]	☑ Y [mm]
1	1	0,	0,
2	1	1,	-9,
3	2	2,	-18,
…			
7	6	6,	-54,
8	7	7,	-63,
*			

a. *b.* *c.*

C.6. *Loads modelling*

Obs. Since the pressing force is unknown, it can be considered that the displacement imposed on it as a constraint (see subchapter above) is an external load of unknown value to be determined by this analysis.

D. SOLVING THE AEF MODEL

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

, Outline: → ↵ ⊞ Solution (A6) → ↵ Solution Information, Details of "Solution Information",
↵ ⊞ Solution Information: ↵ Solution Output → [select from list with ↵, ↵ Force Convergence] (the criterion of force convergence is adopted).

D.2 Setting the results

Selecting the total displacements

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

Setting the displacement according to the Y direction

↳ Solution (A6) → ↵ Insert → ↵ Deformation → ↵ Directional; Details of "Directional Deformation", Definition:
↵ Orientation → [select from list with ↵, ↵ Y Axis];

Setting the equivalent stress

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

Setting the structural error

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

Setting the reaction force (in the displaced area)

↳ Solution (A6) → ↵ Insert → ↵ Probe → ↵ Force Reaction → Details of "Force Reaction":
↵ ⊞ Definition → ↵ Boundary Condition, [select from list with ↵, ↵ Displacement];
↵ ⊟ Options → ↵ Result Selection → [select from list with ↵, ↵ Y Axis].

Setting the parameters in the contact

↳ Solution (A6) → ↵ Insert → ↵ Contact Tool → ↵ Contact Tool;
↳ Contact Tool → ↵ Insert → ↵ Status;
↳ Contact Tool → ↵ Insert → ↵ Pressure;
↳ Contact Tool → ↵ Insert → ↵ Sliding Distance;
↳ Contact Tool → ↵ Insert → ↵ Gap.

D.3 Launching the solving module

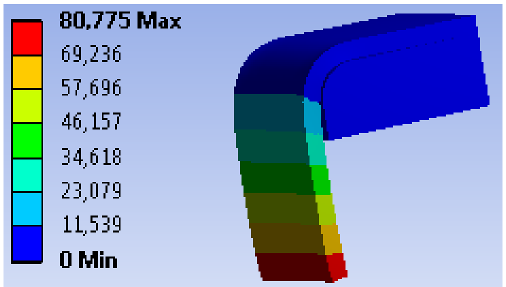
, Outline: ↵ ⊞ Solution (A6) → ↵ Solve.

E. POST-PROCESSING OF RESULTS

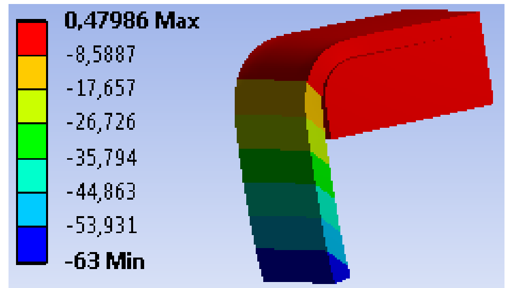
E.1. Viewing the displacement field (total and Y axis)

, Outline: ↵ ⊞ Solution (A6) → ↵ Total Deformation (fig. a); ↵ Directional Deformation (fig. b);
↵ ISO (axonometric visualization); ↵ → [select from list with ↵, ↵ Smooth Contours] (); ↵ → [select from list with ↵, ↵ Show Undeformed WireFrame] (visualization of the non-deformed wireframe structure). ↵ Result → [select from list with ↵, ↵ 1.0 (True Scale)] (selecting the scaling factor); Graph → ↵

Animation (view animation).



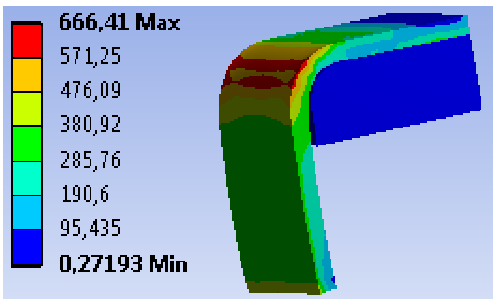
a.



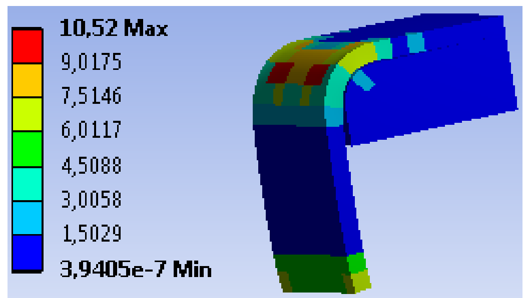
b.

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

M, Outline: ↵ ⊞ ?Solution (A6) → ↵ Equivalent Stress (fig. a); ↵ Structural Error (fig. b).



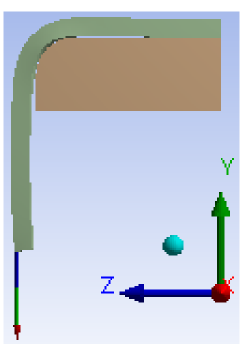
a.



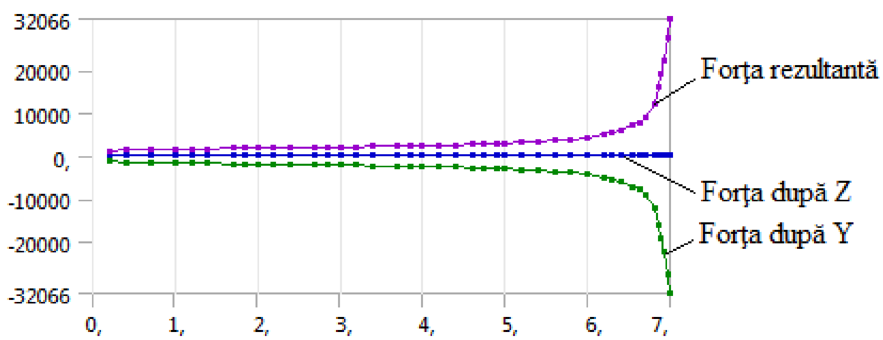
b.

E.3. Visualization of the reaction force

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



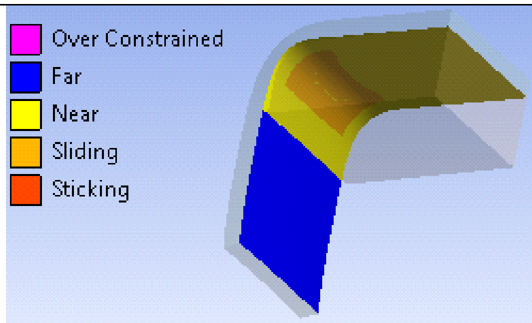
a.

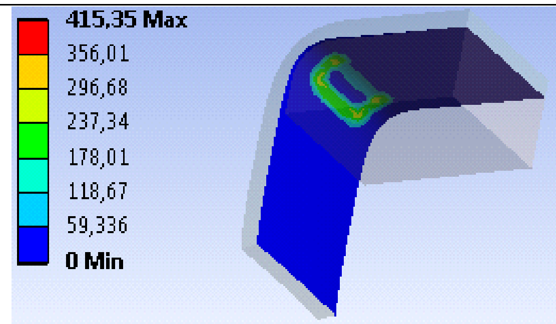


b.

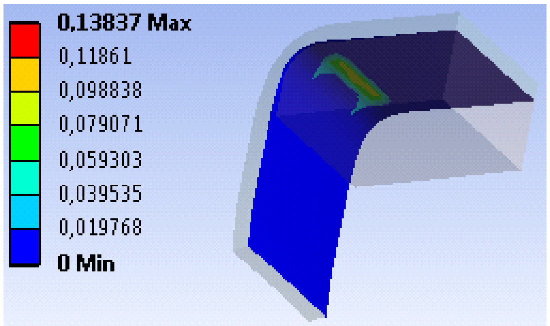
E.4 Viewing the fields of the contact parameters (state, pressure, slip, jump)

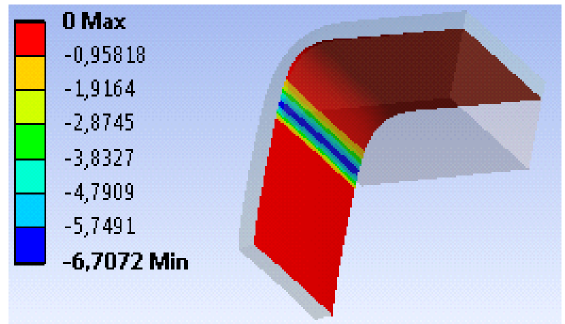
M, Outline: ↵ ⊞ ?Solution (A6) → ↵ ⊞ Contact Tool → ↵ Status (fig. a); ↵ Pressure (fig. b); ↵ Sliding Distance (fig. c); ↵ Gap (fig. d).





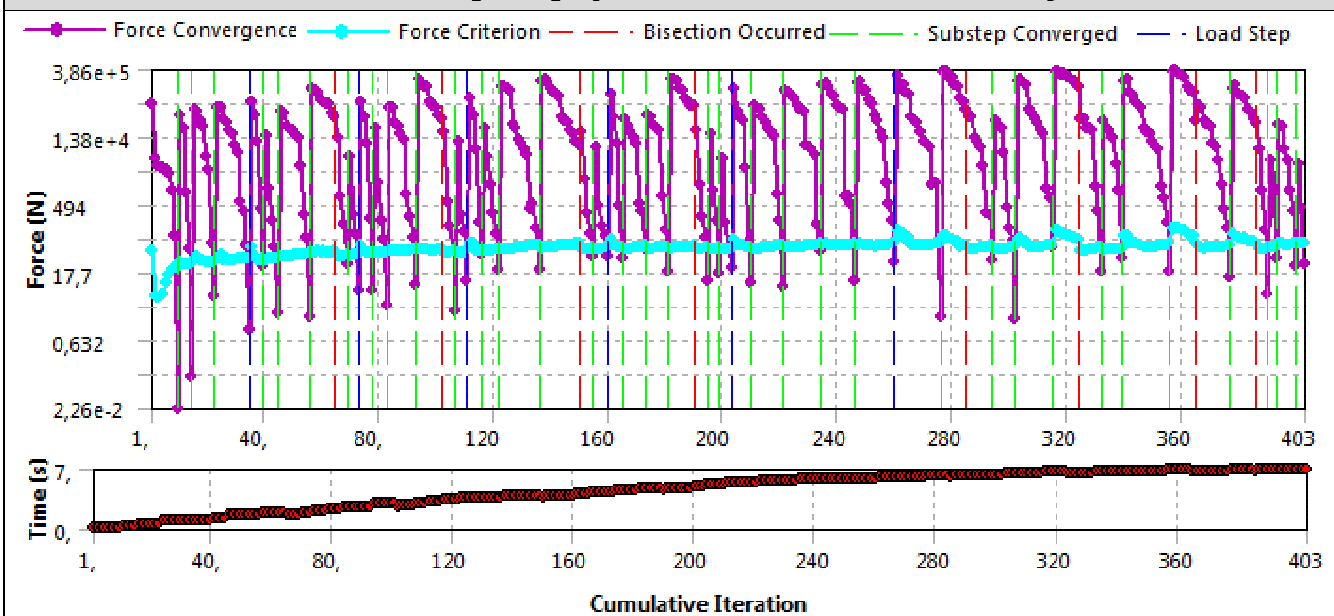
a. *b.*





c. *d.*

E.5 View the convergence graphs of the solution of the nonlinear problem



F. ANALYSIS OF RESULTS

F.1 Interpretation of results

Following the analysis of the results obtained as a result of the modeling and AEF (subchapter E) the following are highlighted:

- Following the deformation process of the semi-finished product as a result of the action of the punch it is observed that the *wings are curved* (subchapter E.1); the maximum total displacement is 80,776 mm (subchapter E.1 fig.a); the displacement in the X-axis direction is 63 mm (subchapter E.1 fig.b), the same value imposed as constraint.
- The maximum equivalent stress has the value of 666.35 MPa (subchapter E.2, fig. A) in the outer curved area of the semi-manufactured greater than the flow tension (295 MPa, subchapter B.3) indicates the *existence of the plastic flow process*. On the other hand, the value of the maximum equivalent stress

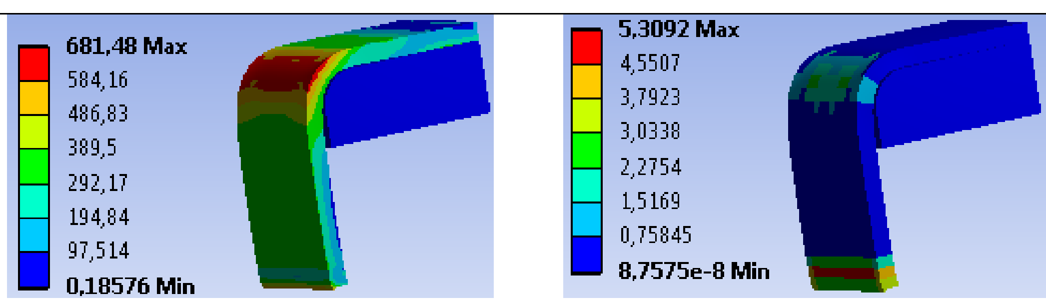
(666.35 MPa) being higher than the breaking stress of the material (max. 660 MPa, subchapter B.3) highlights the possibility of *breaking cracks* (subchapter F.3)

- The variation of the interaction force, increasing up to 32094 N, between the punch and the blank during the plastic deformation process is presented in subheading. E.3, fig. b. The values increased in the last part of the deformation process, situation shows that the value of the imposed displacement is greater than the real one and it is necessary to repeat the analysis with smaller values (eg 62.8 mm); the maximum value of the reaction force is the basis of the deformation device calculation.
- In the subcap. E.5 the contact states are visualized (subchapter E.5, fig. A) and the values of some contact parameters: pressure - max 415.35 MPa in the connection area, fig. b; relative slip - max 0.13837 mm in the upper area of the connection; play (jump) - max 6.7072 mm in the lower area of the connection. These values are useful for designing the workpiece and the mold. For example: starting from the maximum pressure value, the hardness of the active surface of the mold and the level of crushing of the semi-finished material inside the connection is determined; starting from the observation that the deformed wing of the blank is curved (subchapter E.4, fig. a; unwanted shape) and that the clearance between the die and the blank is increased (6.7072 mm) it is emphasized that the shape of the punch must be changed such as this to press on the semi-finished product and in the connection area a case involving the remodeling of the problem (subchapter F.3, fig.d)

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 (subchapters E.3 and E.6) the following are highlighted:

- The maximum structural error in the outer zone of the connection with increased value (10.52 mJ) shows increased deviations from the quasi-exact value (unknown). To reduce deviations, the fineness of discretization is increased and the model is solved. Thus, by changing the size of the finite element by scrolling through the sequence: ↵ ⊞ Mesh → ↵ Face Sizing / Face Sizing 2 → Details of "Face Sizing" - Sizing / Details of "Face Sizing 2" - Sizing, ↵ ⊞ Definition : ↵ Element Size → [enter the value, -3]. After solving (M; ↵ Solve) the maximum reduced structural error, 5.3092 mJ, is obtained in the imposed displacement area (fig. B); the fact that in the area with the maximum equivalent stress (681, 48 MPa, fig. a) the values of the structural error are reduced (approx. 2… 3 mJ), it shows that the equivalent stress is very close to the quasi-exact one.
- The model solution convergence is done in 403 steps (subchapter E.6) and the computation time is increased.



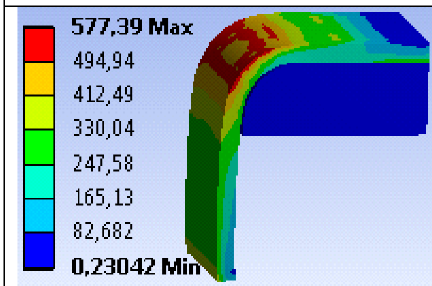
a. ***b.***

F.3 Design studies

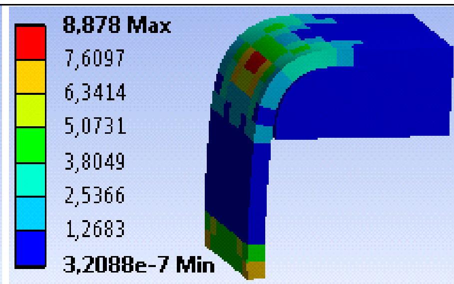
In order to avoid the occurrence of the rupture micro-cracks in the external connection area of the semi-finished product (subchapter F.1) it is necessary to reduce the maximum equivalent stress; In this case, the connection radius and / or the decrease of the plate thickness can be adopted, in compliance with the constructive-functional requirements. Thus, to increase the connection radius to the value of 15 mm, it is necessary to modify the analysis model and to solve the model by going through the sequences:

: DM, Tree Outline: ↵ Extrude2 → ↵ FBlend1 → Details View, **Details of FBlend1**: ↵ FD1, Radius (>0) → [input value, -15]; ↵ Generate; Outline: ↳ Geometry → ↵ Refresh Geometry; ↵ Connections → ↵ Contacts → ↳ Contact Region → ↵ Delete (option Contact Region automatically apear with Refresh Geometry and because no other contacts are inserted, it is deleted); ↵ Displacement → Tabular Data: [change the value in step 7, -65 (instead of -63)]; ↵ Solve.

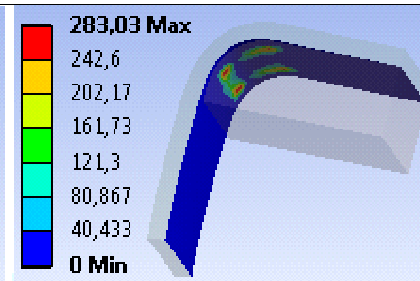
After solving, reduced values of the maximum equivalent stress (577.39 MPa, fig. a) and the contact pressure (283.03 MPa, fig. c) corresponding to the structural error of 8,878 mJ are obtained.



a.

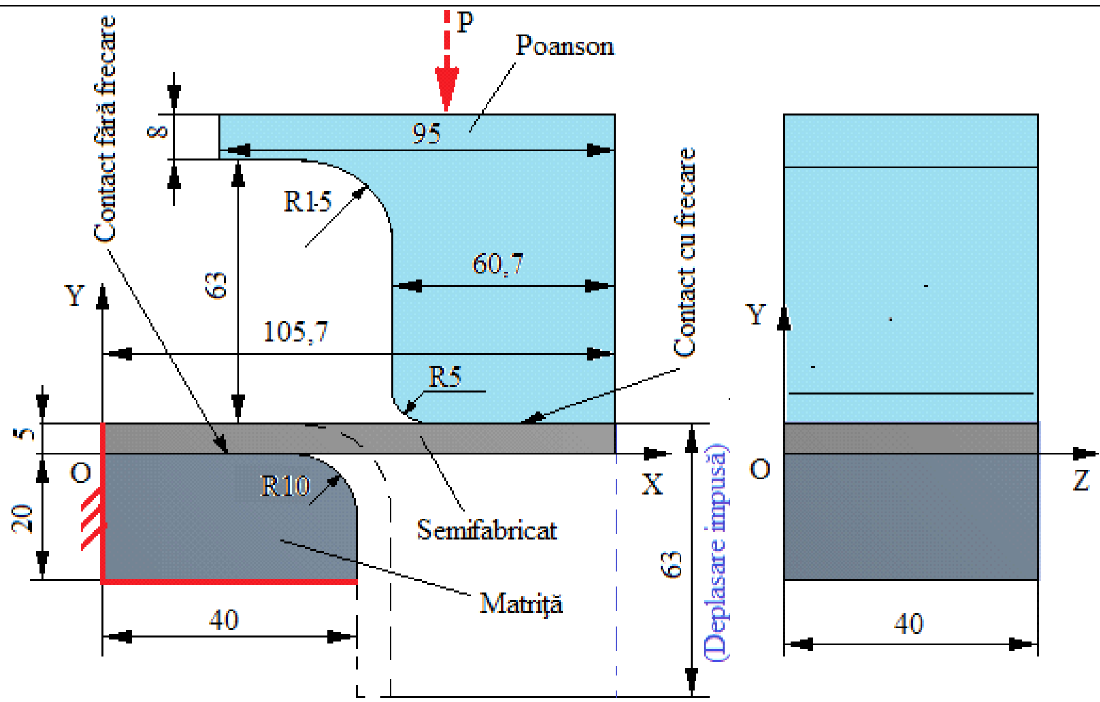


b.



c.

If the design requirements require small deviations of the radius and linearity of the wings of the profile obtained from the imposed values, it is necessary that the bending device contain a punch with a contour that "forces" the plastic deformation of the blank to follow the contour of the mold (fig. d). Thus, the analysis model will have a third solid (Poanson) that will be in contact with the slip friction with the Semifabricat object (μ = 0.2). For AEF, as an exercise in this application, the same material as the mold will be adopted for the punch.



d.

G. CONCLUSIONS

In this paper, the modeling and the 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 AEF in ANSYS Workbench, which insists, especially, on the modeling and analysis of a deformable element in the plastic deformable area applying large displacements imposed.
The adopted AEF model involves considering the frictionless contact between two elements as well as a material with nonlinear behavior. The deformation force being unknown, the imposed displacement of the edge of the blank is introduced as loading.
As a result of solving the nonlinear model with finite elements adopting the method of force convergence, we obtained results with increased precision, the values of the obtained parameters (displacements, tensions, force) being useful for the design of the workpiece as well as of the bending device.