Application: AEF-A.2
Cantilever beam with fillet

KEY WORDS
Static linear analysis, Planar geometric model, Plane stress state, Linear material, Planar 2D finite element, Linear finite element, Parabolic finite element, Machine element, Checking with classical models, Cantilever beam, Singularities

CONTENT

A. PROBLEM DESCRIPTION

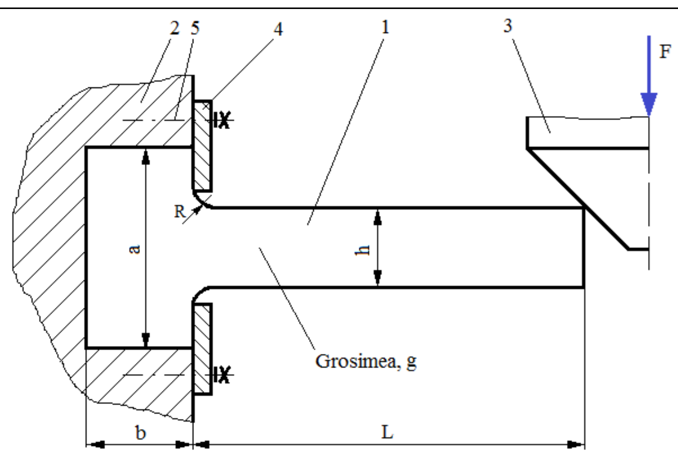
A.1 Introduction

In many training situations to solve AEF problems, especially as a beginner or initiating a new AEF platform, it is recommended to solve simple problems that are reduced to classic models with known analytical solutions but also by highlighting the not recommended extreme situations (eg singularities) in the current practical applications.

The main objective of this application involves the development of an AEF for a rectangular beam embedded on a contour, comparing the results with the classical analytical ones and highlighting the effects of the singularities involved by the theoretical cases of concentration of tensions in the zones of fillet with zero radius and in the zones of action of the concentrated forces on reduced surfaces at a point or a line.

A.2 Application description

In the structure of the support device below, the elastic support element 1, firmly positioned in the body 2 through the bars 4 and the screws 5, must ensure a displacement imposed under the action of the press force F, developed by the slide 3, and return to the initial state after its cancellation.



A.3 The application goal
In this application, the analysis of the displacement, deformation and tension fields of the support element 1 made of C55 steel and with the following dimensions is followed: L = 100, h = 10 mm, g = 10 mm, a = 50 mm, b = 20 mm . Starting from the fact that the element 1 has a constant constant thickness and the load with F = 1000 N is uniformly produced in width, it is emphasized that the problem is classified in the plane state of stresses (the voltages are invariable in thickness) and, therefore, the analysis with finite elements will be make customizations for this case. In addition, compared to the analysis from the application of AEF-A.1 the effects of the singularities of concentration of tensions in the "sharp" (null radius) and action areas of the "needle" (point) or "knife" (on the line) forces will be studied.

B. THE FEA MODEL

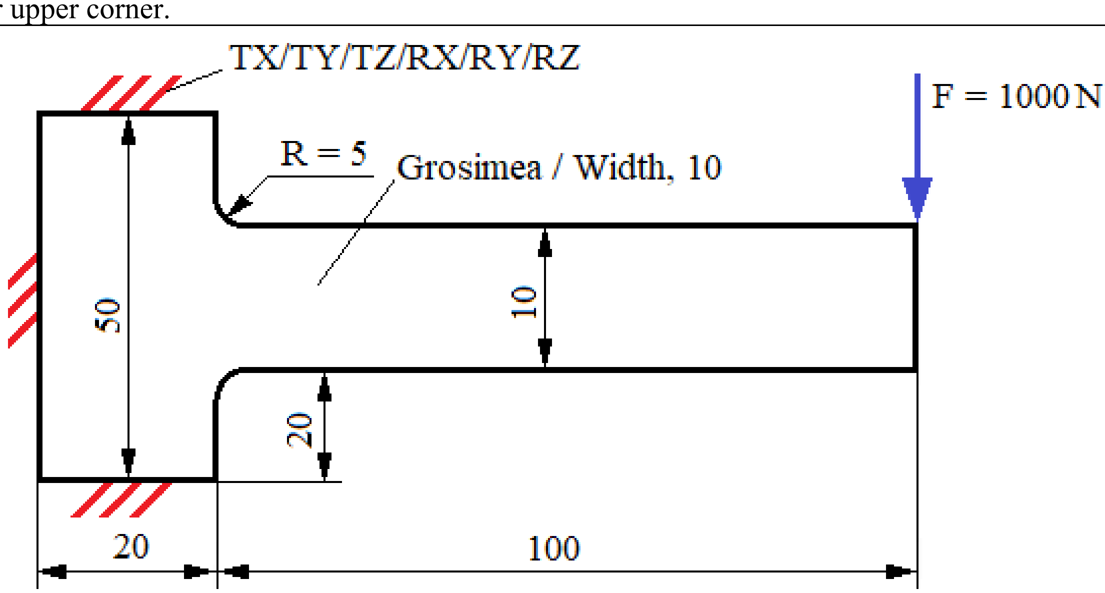
B.1 The model definition

In order to compare the results obtained by the finite element analysis with the classical solution model based on the material resistance methods (embedded beam), the most simplified possible model is adopted which implies:

- simple geometric shape,
- adoption of material strengthens constraints (embedding)
- the loads are concentrated,
- the material has a linear behavior

B.2 The analysis model description

Because the structure can be included in the AEF plane stress state, it can be modelled in plane, considering the rectangular geometric shape 100 mm long and 10 mm wide with 2D finite elements.
Geometric constraints involving cancellations of translation and rotation relative to the X, Y and Z-axis, respectively, apply to the points on the Y-axis edge. Load the model with the concentrated force F = 1000 N in the far upper corner.



B.3 Choosing the characteristics of the material and the environment

For AEF, the strength characteristics of the C55 steel are:

- longitudinal modulus of elasticity, $E = 210000\ N / mm^2$;
- Poisson's ratio, $\nu = 0{,}3$.

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 ↵ X].

Saving of the project

↵ Save As... → Save As, File name: [enter name, AEF-A.2] → ↵ 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, 200000] → ↵ 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)

→ ↵ Sketching → (Look At Face/Plane/Sketch) [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) → [drawing two rectangular lines with the pencil indicator marking with, ↵ from a point of Y axis (C symbol appear), and finish in opposite point simultaneously with release of the mouse ↵] (fig. b).

Outline beam generation

↵Modify → ↵ Trim → [it will be deleted by selecting with ↵ the portions of the straight segments that do not belong to the contour (fig. c)].

Center lines in relation to the X axis

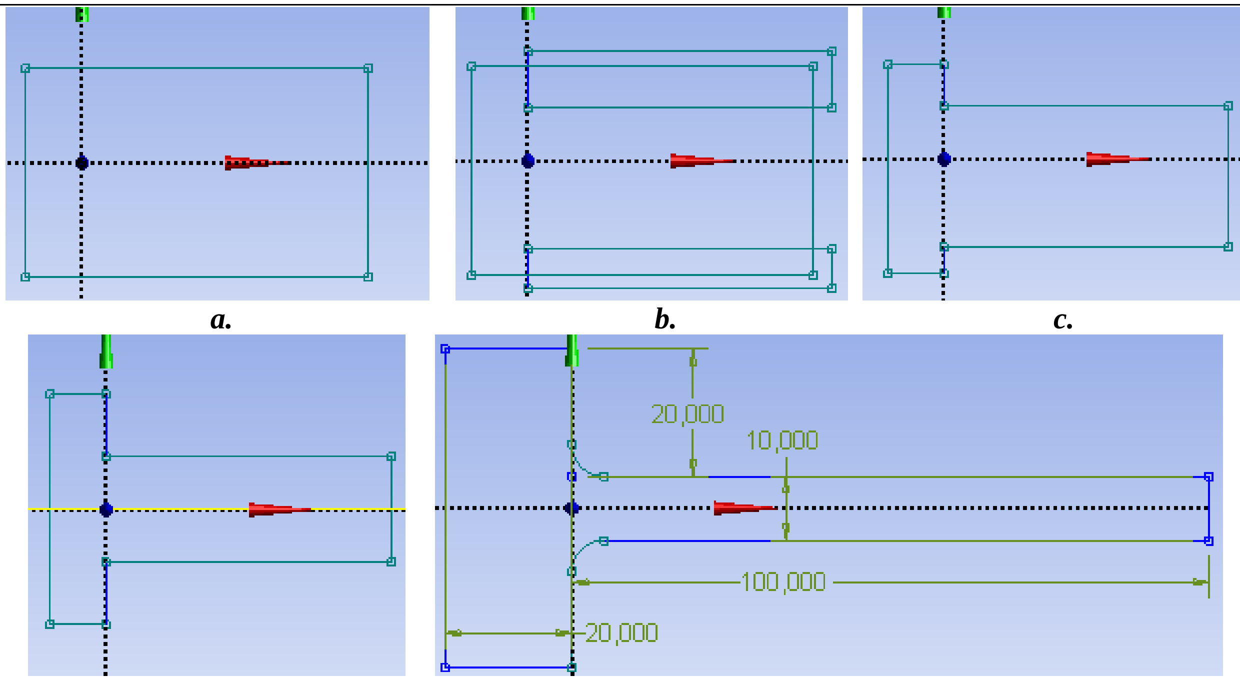
↵ Constraints →↵ Symmetry → [select with ↵ the X axis and then the two parallel lines with this axis to the left of the Y axis (fig. d)] → [select with ↵ the X axis and then the two parallel lines with this axis to the right of the Y axis (fig. d)].

Dimensions

↵Dimensions → ↵ Semi-Automatic → [dimensions are automatically activated with ↵] → Details View, ⊟ Dimensions: 4: → [they are inserted into the boxes ☐ L1, ☐ L2, ☐ L3, ☐ L4 (fig. e)]. ↵ Display (viewing dimensions), Name: ↵☐ (it is disabled), Value: ↵☑ (is activated). ↵ Move (moving dimensions), [the dimension activates with ↵ and moves keeping the activation to the desired position] (fig. e).

Fillet generation

↵Modify → ↵ Fillet → [input Radius:, radius value, 5] → [select with ↵ the connecting lines (fig. e)]

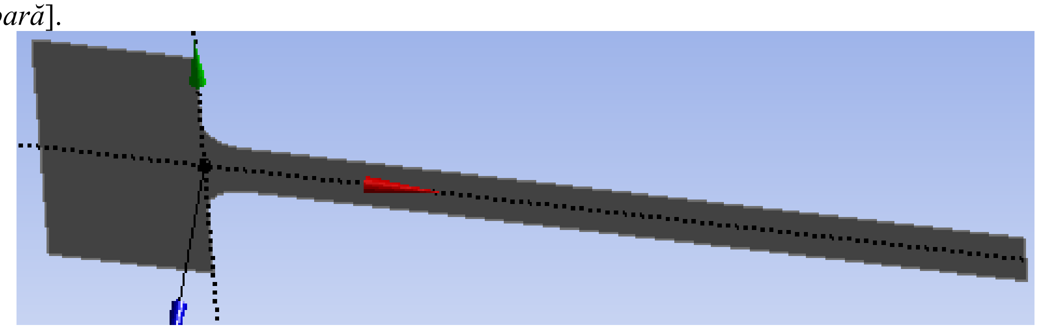


a. **b.** **c.**

d. **e.**

C.3.3 Surface generation

→ Concept → Surfaces From Sketches → Details View, Details of SurfaceSk1 : Base Objects → Tree Outline → XYPlane, Sketch1 → Apply ; Thickness (>=0), [input thickness, 10] → Generate
.
Sketch1 → Hide Sketch (hide sketch) (axonometric visualization).
1 Part, 1 Body → Surface Body → Details View, Details of Surface Body : Body, [input name, *Suprafață bară*].



C.3.4 Save of geometric model

→ (Save Project) → (Close Design Modeler).

C.4. Finite element modelling

C4.1 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 → Edit... → [launching the *Mechanical [ANSYS Multiphysics*].

Setting the type of the problem

→ Outline → Geometry → Details of "Geometry", Definition : 2D Behavior, [select from list , Plane Stress (default settings)].

Setting the material characteristics

Outline → Geometry → Suprafață bară → Details of "Suprafață bară" : Material : Assignment, [is selected from the list , Structural Steel] (usually, when there is only one material, this setting is default).

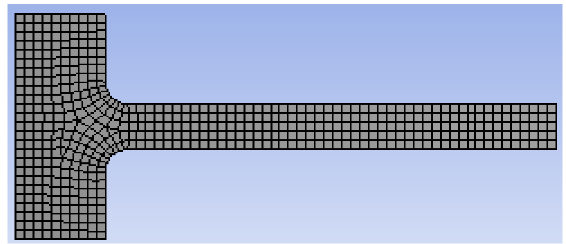
Setting the units

→ Units → Metric (mm, kg, N, s, mV, mA).

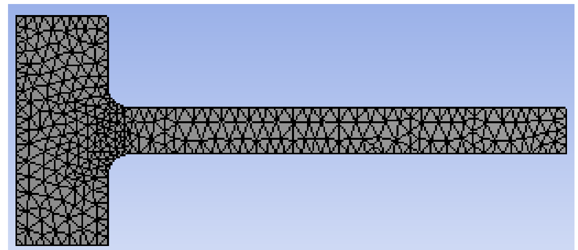
C.4.2 Model meshing

<u>*Setting general meshing features*</u>

→ Outline → ↵ Mesh → Details of "Mesh" → ⊞ Sizing : Use Advanced Size Function, [is selected from the list with ↵▼, ↵ Off]; Relevance Center, Use Advanced Size Function, [selecting from the list with ↵▼, ↵Fine] ; ☐ Element Size, [enter the dimension of the finite element: 4, 2 (fig. a, b), 1, 0.5] (for each of the two cases below, 4 analyzes will be performed one for each value) → ⊞ Defaults : ☐ Relevance , [it is generated with ↵, valoarea / value 100] → ⊞ Advanced : Element Midside Nodes, [selecting from the list ↵▼, ↵Dropped/Kept] (for each finite element dimension there will be 2 analyzes, one with linear finite element (without intermediate nodes on sides, variant Kept) and parabolic finite element with one intermediate node on sides, variant Kept).



a.



b.

<u>*Selecting triangular finit element (case I) or rectangular (case II)*</u>

↳ Mesh → Insert → Method → Details of "Automatic Method" - Method → ⊞ Scope : Geometry, [selecting with ↵ geometrical model] → ↵No Selection, ↵ Apply → ⊞ Definition : Method , [selecting from the list ↵▼, ↵Triangles/Quadrilateral Dominant].

<u>*Creating the model with finite elements*</u>

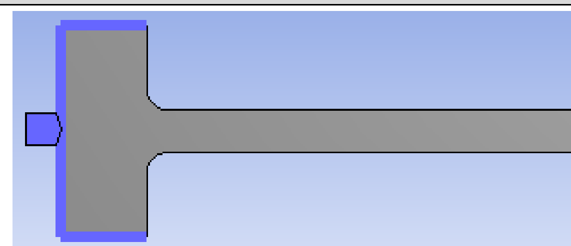
↵ Mesh → Generate Mesh (fig a,b).

<u>*View the meshing statistics*</u>

↵ Mesh → Details of "Mesh", ⊞ Statistics : ☐ Nodes , 422/605; ☐ Elements , 664/516.

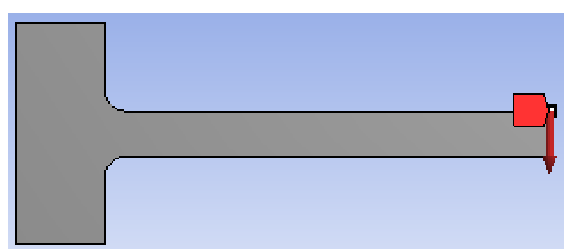
C.4.3 Supports and restraints modelling

→ Outline → ↵ Static Structural (A5) → ↵ Supports → ↵ Fixed Support → Details of "Fixed Support", Geometry → ↵ (activating line selection filter) → [selecting with Ctrl+↵ clamp edges] → ↵ Apply.



C.4.4 Loads modelling

→ Outline → ↵ Static Structural (A5) ↵ Loads → ↵ Force →Details of "Force", ⊞ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ peak] → ↵ Apply; ⊟ Definition : Define By , [selecting from the list ↵ ▼, ↵Components], ☐ Y Component, ↵0, N (ramped) → [input value -1000].



C.4.5 Saving the project

→ ↵ File → Save Project...

D. SOLVING THE AEF MODEL

D.1 Selecting the results

Selecting the total displacements

→ Outline → ↵ Solution (A6) → ↵ Deformation → ↵ Total.

Selecting the stress fields

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

Selecting the structural error

↵ Solution (A6) → ↵ Stress → ↵ Error.

Selection of the equivalent voltage field along a line from the connection base (fig. a)

Generation of Line 1: Model (A4) → ↵ Construction Geometry → ↵ Path → Details of "Path", Start: Start X Coordinate, [input value 5]; Start Y Coordinate, [input value, 5], (the coordinates of the point of upper fillet base, fig. a) → End X Coordinate, [input value, 5]; End Y Coordinate [input value, -5] (the coordinates of the point of lower fillet base, fig. b) .

Selecting the field of equivalent stress after Line 1:

↵ Solution (A6) → ↵ Stress → ↵ Equivalent (von-Mises) → Details of "Equivalent Stress 2" → ↵ Scope: ↵ Scoping Method, [selecting from the list ↵, ↵ Path]; Path, [selecting from the list ↵, ↵ Path].

Selecting the equivalent stress field after Line 1 (fig. b)

Generation of Line 2: Model (A4) → ↵ Construction Geometry → ↵ Path → Details of "Path", Start: Start X Coordinate, [input value 100]; Start Y Coordinate, [input value, 5], (the coordinates of the upper point, fig. b) → End X Coordinate, [input value, 100]; End Y Coordinate [input value, -5]; (coordinates of the lower point, fig. b).

Selecting the equivalent stress field after Line 2:

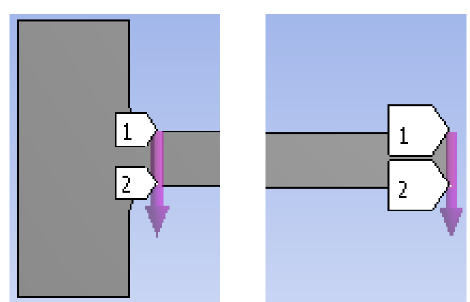
↵ Solution (A6) → ↵ Stress → ↵ Equivalent (von-Mises) → Details of "Equivalent Stress 3" → ↵ Scope: ↵ Scoping Method, [selecting from the list with ↵, ↵ Path]; Path, [selecting from the list with ↵, ↵ Path 2].

Selecting the field of structural error after Line 1 (fig. a)

↵ Solution (A6) → ↵ Stress → Error → Details of "Structural Error" → ↵ Scope: ↵ Scoping Method, [selecting from the list ↵, ↵ Path]; Path, [selecting from the list ↵, ↵ Path].

Selecting the field of structural error after Line 2 (fig. b)

↵ Solution (A6) → ↵ Stress → Error → Details of "Structural Error 2" → ↵ Scope: ↵ Scoping Method, [selecting from the list ↵, ↵ Path]; Path, [selecting from the list ↵, ↵ Path 2].



a. **b.**

D.2 Launching the solving module

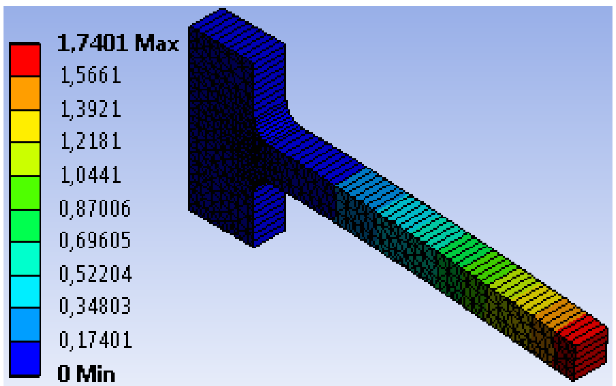
→ ↵ Solution (A6) → ↵ Solve.

E. POST-PROCESSING OF RESULTS

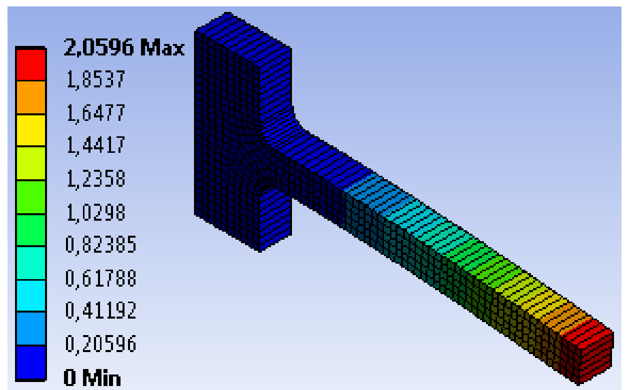
E.1 Viewing the total displacement field

→ Outline → ↵ Solution (A6) →↵ Total Deformation (fig. a, for Case I fig. b, for Case II); ↵ ISO (axonometric visualization); ↵ → [selecting from the list ↵, ↵ Smooth Contours] (visualization of smooth contours) ; ↵ → [selecting from the list with ↵, ↵ Show Elements] (visualization of finite elements) ; ↵ Result → [selecting from the list with ↵, ↵ 3, (Auto Scale)] (selecting the scale of displacement); ↵ MAX (marking the node with the maximum total displacement); ↵ MIN (marking the node with the minimum total displacement).

Graph → ↵ Animation (visualisation af animation).



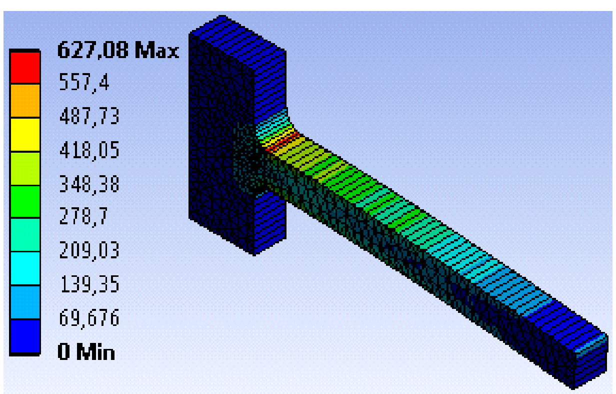
a.



b.

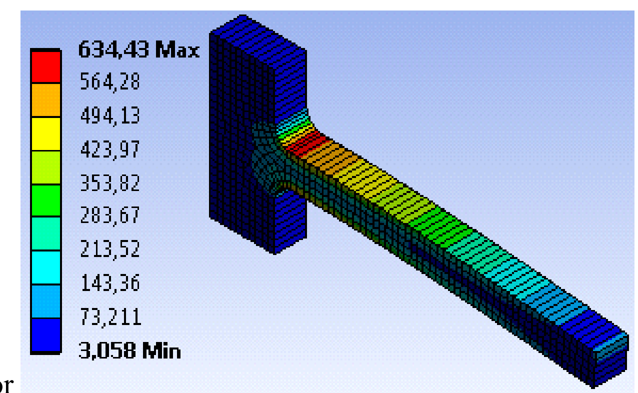
E.2 *Viewing equivalent stress fields*

→ Outline → ↵ Solution (A6) →↵ Equivalent Stress (fig. a, for **case I**; fig. b, for **case II**);

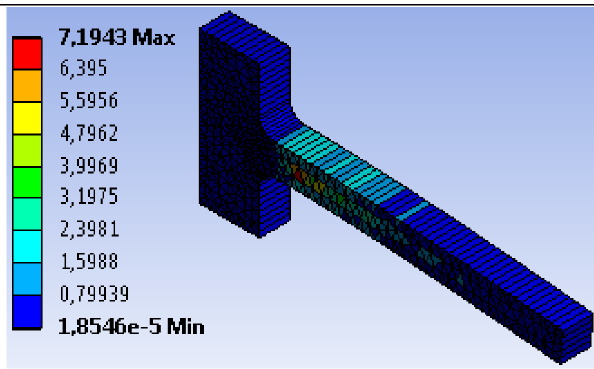
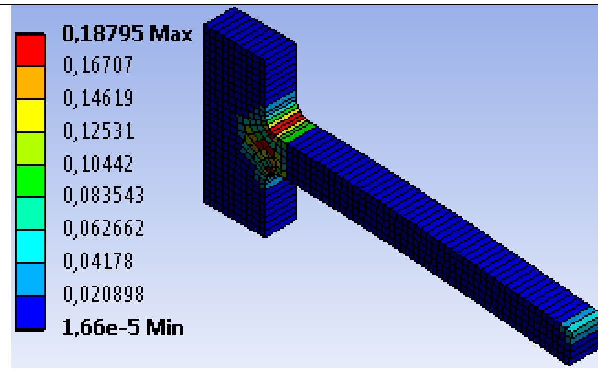


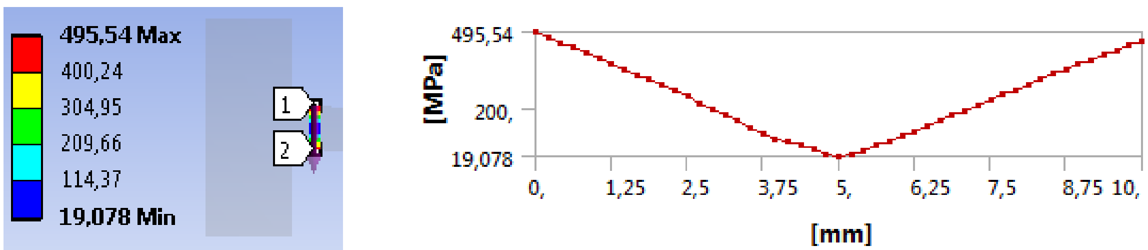
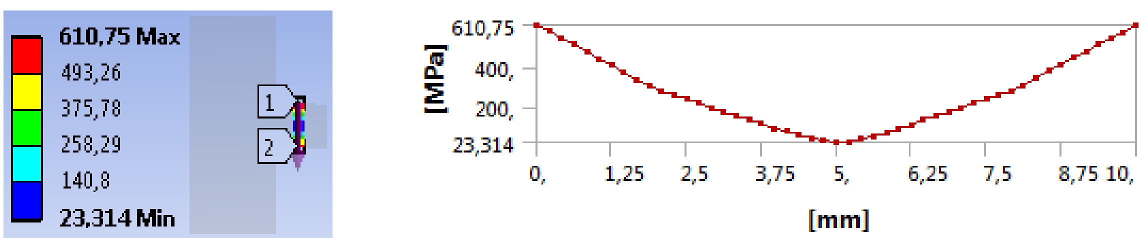
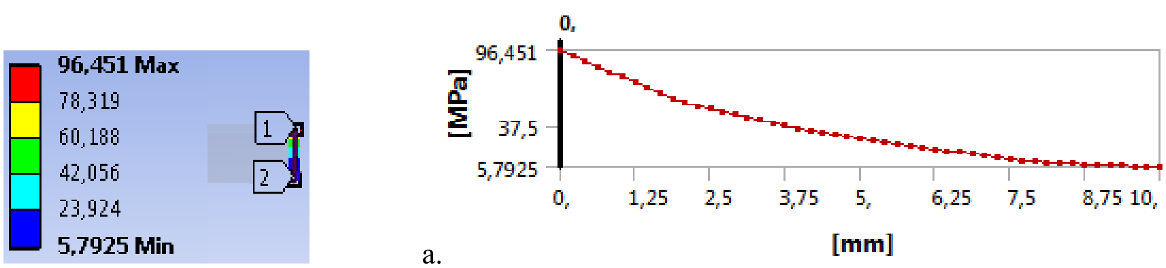
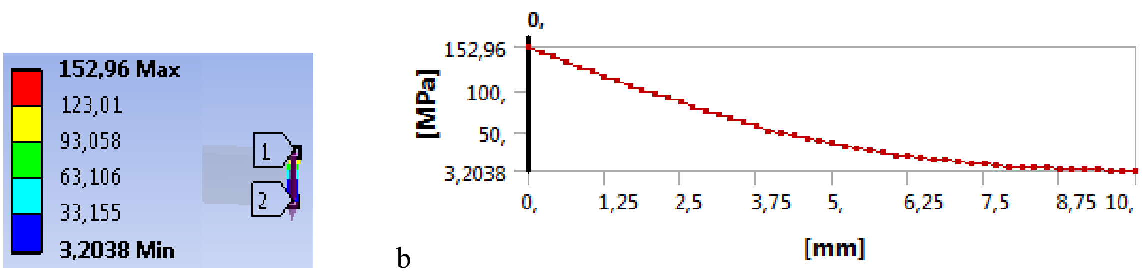
a.

for



b.

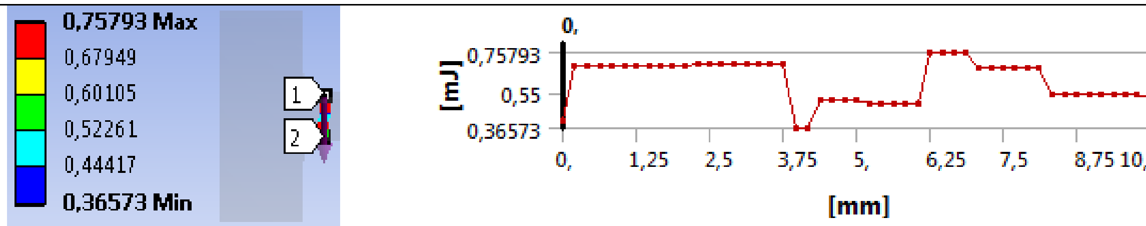
E.3 *Viewing structural error fields*	
Outline → Solution (A6) → Structural Error (fig. a, for case I; fig. b, for case II);	
 ***a.***	 ***b.***

E.3 *Viewing equivalent stress fields on line*
E.3.1 Viewing equivalent stress fields on Line1
Outline → Solution (A6) → Equivalent Stress 2 (fig. a, for **case I**; fig. b, for **case II**);
 ***a.***
 ***b.***
E.3.2 Viewing equivalent stress fields on Line2
Outline → Solution (A6) → Equivalent Stress 3 (fig. a, for **case I**; fig. b, for **case II**);
 a. ***a.***
 b

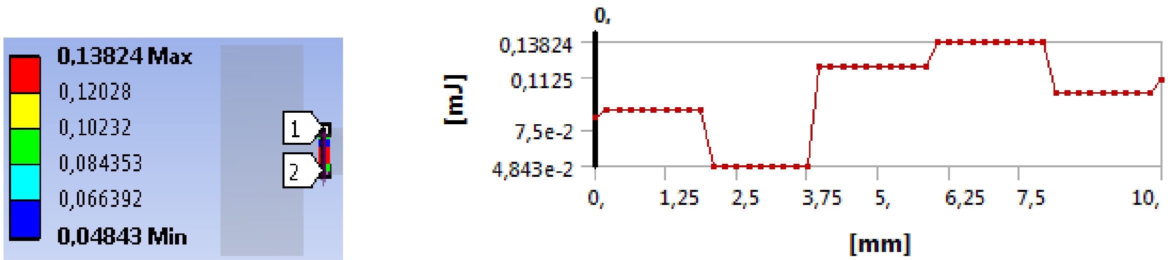
E.4 *Viewing structural error fields*

E.4.1 Viewing structural error fields on Line 1

→ Outline → Solution (A6) → Structural Error 2 (fig. a, for **case I**; fig. b, for **case II**);



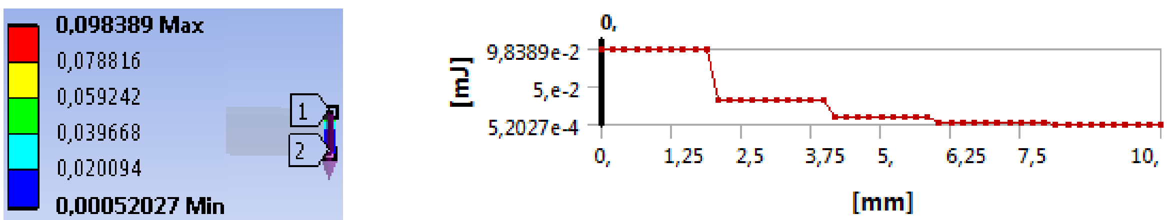
a.



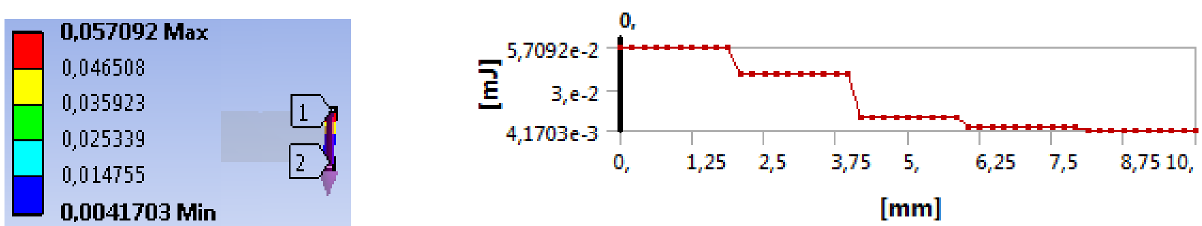
b.

E.4.2 Viewing structural error fields on Line 2

→ Outline → Solution (A6) → Structural Error 3 (fig. a, for **case I**; fig. b, for **case II**);



a.



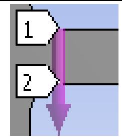
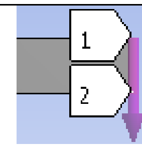
b.

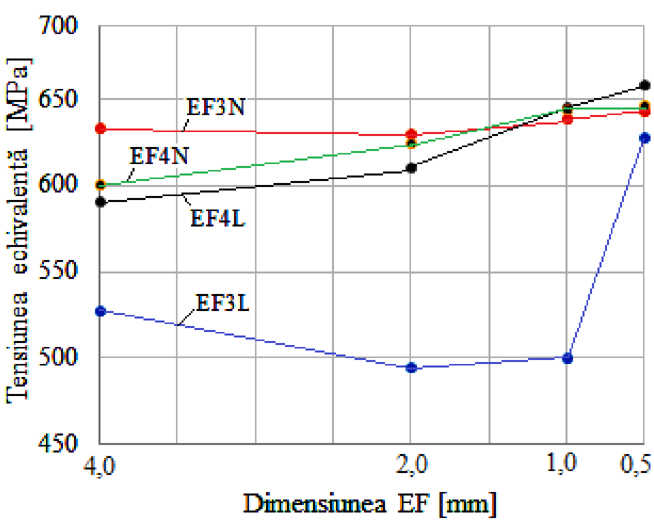
F. RESULTS ANALYSIS

F.1 Summary of analysis results

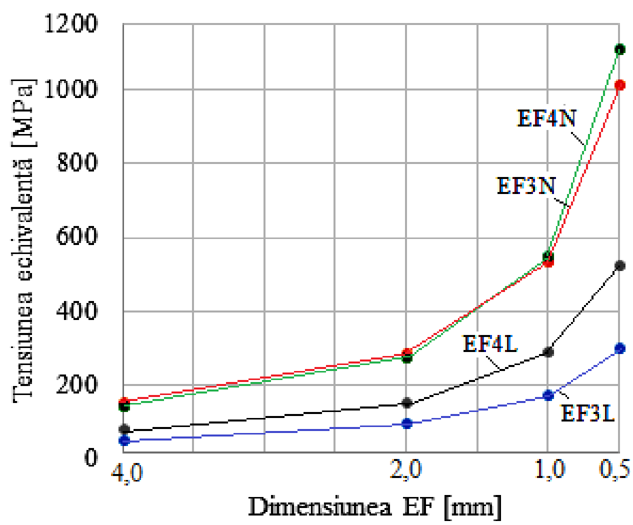
In order to highlight aspects related to the accuracy of the results and the convergence of the solution, analyzes were performed considering for modeling linear finite elements (without intermediate nodes on sides) and nonlinear (with intermediate nodes on sides), with two forms (triangular and rectangular) and with four dimensions each. The results are summarized in the table and graphs below. In subchapter E, the results for the cases of the linear, triangular and rectangular finite element, with the dimension of 2 mm are presented (analyzes II and VI in the table).

Tab. a

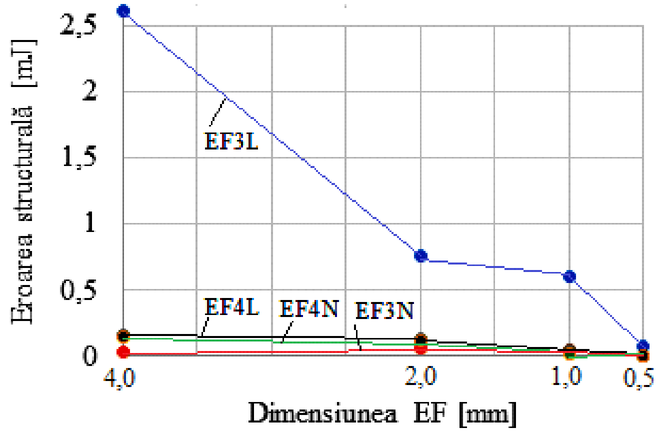
Analysis code	Dimension of finite element	Number of nodes	Number of finite elements	Total maximum displacements	 (Line 1) Equivalent stress	(Line 1) Structural error	 (Line 2) Equivalent stress	(Line 2) Structural error
First order triangular finit element (without intermediate nodes on edges, EL3L)								
I	4	194	284	1,447	527,48	2,619	53,26	0,105
II	2	422	664	1,74	495,54	0,758	96,451	0,098
III	1	1186	2034	1,958	500,08	0,606	176,53	0,087
IV	0,5	4210	774,6	2,026	628,65	0,081	302,03	0,052
Second order triangular finit element (parabolic, with intermediate nodes on edges, EL3N)								
V	4	671	284	2,011	633,91	0,0395	153,05	0,2951
VI	2	1607	664	2,024	629,91	0,0576	285,81	0,2233
VII	1	4405	2034	2,019	638,78	0,0437	537,15	0,1845
VIII	0,5	16165	7746	2,017	643,27	0,0013	1018	0,1695
First order rectangular finit element (without intermediate nodes on edges, EL4L)								
IX	4	230	179	2,054	591,14	0,1606	82,315	0,06
X	2	605	516	2,06	610,75	0,1383	152,96	0,057
XI	1	2169	2001	2,064	645,58	0,0485	291,52	0,055
XII	0,5	8382	8050	2,068	658,97	0,0059	525,04	0,047
Second order rectangular finit element (parabolic, with intermediate nodes on edges, EL4N)								
XIII	4	631	176	2,0366	600,91	0,14	145,61	0,6
XIV	2	1739	520	2,0383	624,74	0,125	275,72	0,4737
XV	1	6340	2001	2,036	644,81	0,0129	551,91	0,4225
XVI	0,5	24781	8038	2,036	646,81	0,0026	1111,14	0,356



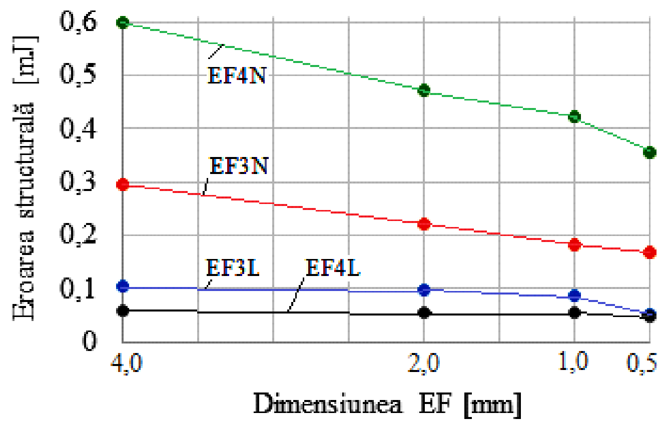
a.



b.



c.



d.

Following the analysis of the results obtained as a result of the modeling and AEF (subchapter E) and the use of the classical computational relations (the application A.1 subchapter F.1) obtained under the conditions of the materials strength hypotheses, the following are highlighted:

- The maximum total displacement for the case of the linear triangular finite element with large dimensions (4, 1 mm) has values (1.447 and 1.74 mm respectively according to lines I and II of the tables a and subchapter E.1, fig. A) with large deviations (-27.65%; -13%) from the value (2 mm) obtained from the theoretical analytical model (the application A.1 subchapter F.1).
- The maximum equivalent voltage in the connection area with voltage concentrator (visualized after a transverse line in the connection area) for the large triangular linear element (EF3l) with large dimensions (4, 2, 1 mm) has values (527.48; 495 , 54 and 500.08 MPa respectively according to lines I, II and III of the table, the graph of Fig. A and subchapter E.3.1, Fig. A) with large deviations (-18.84%; -28.37%; -23.06%) compared to the value of the maximum equivalent convergence voltage (approx. 650 MPa, fig. A); In the case of the other analyzes (cases I and II) - with large dimensions (4, 2 mm) we can see in fig. of much smaller deviations (<9%) of the values of the maximum equivalent voltage compared to the value of the maximum equivalent convergence voltage (approx. 650 MPa, fig. a).
- The maximum equivalent voltage in the head area of the bar (viewed from the force line) has small values (<300MPa, fig. B), for large dimensions (4, 2 mm) of the finite elements, and has much increased values (> 300MPa, fig. B); the value of the equivalent voltage increases exponentially (fig. b) with the increase of the discretization fineness (EF dimensions <0.5 mm) which highlights the singularity effect of the tension consequence of the load with concentrated force - a theoretical situation in which much increased values of the voltage result around the force action point (tensions in this area are not taken into consideration when designing).

F.2 Analysis of convergence and precision

Following the analysis of the tensile values of the structural errors (tab. A, fig. C, d) the following are highlighted:

- In fig. c, table. a, but also in subchapter. E.4.1, fig. to highlight the variation of the structural error with the increase of the discretization fineness. The increased values of the structural error (> 0.5 mJ) for the linear triangular finite element (EF3L) with enlarged dimensions (> 1 mm) in correlation with the variation of the equivalent voltage in fig. to also highlight in this way the increased deviations of the voltages from the quasi-real value (approx. 650 MPa, fig. a). For the other finite elements (EF4L, EF4N and EF3N) we observe reduced values of structural error (<0.16 mJ) and for large finite element dimensions (4, 2 mm) that decrease, in correlation with the increase (convergence) of the stresses. equivalent (fig. a), with the increase of the discretization fineness (decrease of the finite element size).

- In fig. d we observe reduced values of the structural error (<0.6 mJ, for EF3N) and their decreasing variations, which is not correlated with the exponential increase of the maximum voltages (fig. b), thus highlighting the singularity (non-convergence) of the voltage. .
- In fig. to highlight the quasi-exact equivalent voltage (approx. 650 MPa, fig. a) as a result of the convergence of the solution (asymptotically approximating a quasi-exact value for finite elements of different shapes as the fineness of discretization increases (EF dimension decreases).

G. CONCLUSIONS

The modeling and analysis with finite elements of this paper were done more with didactic purpose aiming, on the one hand, the initiation of the user with the main stages of developing an application of AEF in ANSYS Workbench and, on the other hand, the comparison. and evaluating the results obtained from AEF with different shapes and sizes of finite elements.

The adopted AEF model leads to coarse deviations from the exact solution for the linear triangular finite element as opposed to the quadratic finite element model which shows a convergence with very small deviations.

The analysis of the results, in particular, of the tensions, for discretizations with increased fineness, shows that in the area with the singularity of the tension (the point of application of the concentrated force), although the structural error decreases to allowable values which would show a good accuracy, the values of the tensions do not converge towards the cvsiexact value, but they grow non-asymptotically (values that do not correspond in reality).

The AEF model studied in this paper is efficient in terms of modeling possibilities offered by the ANSYS platform, especially for the quadrilateral finite element that ensures good convergence in the connection area (voltage concentrator, real case) and leads to increased errors. , non-convergence of the solution for the force action area is concentrated in one point (theoretical case), a case not recommended in the design practice that can be avoided by considering the force distributed on a line very close to reality.