Application: AEF-A.3
Cantilever beam with singularities

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
B. THE FEA MODEL
C. PREPROCESSING OF THE FEA MODEL
D. SOLVING THE FEA MODEL
E. POSTPROCESING OF THE RESULTS
F. ANALYZING OF THE RESULTS
G. CONCLUSIONS

A. PROBLEM DESCRIPTION

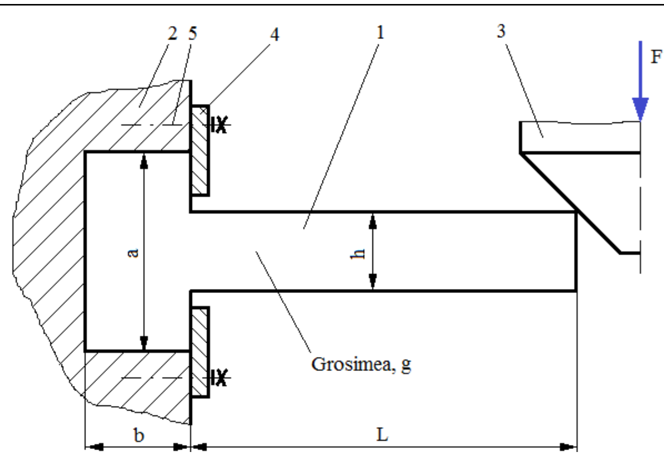
A.1 Introduction

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 (e.g. 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 stresss 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

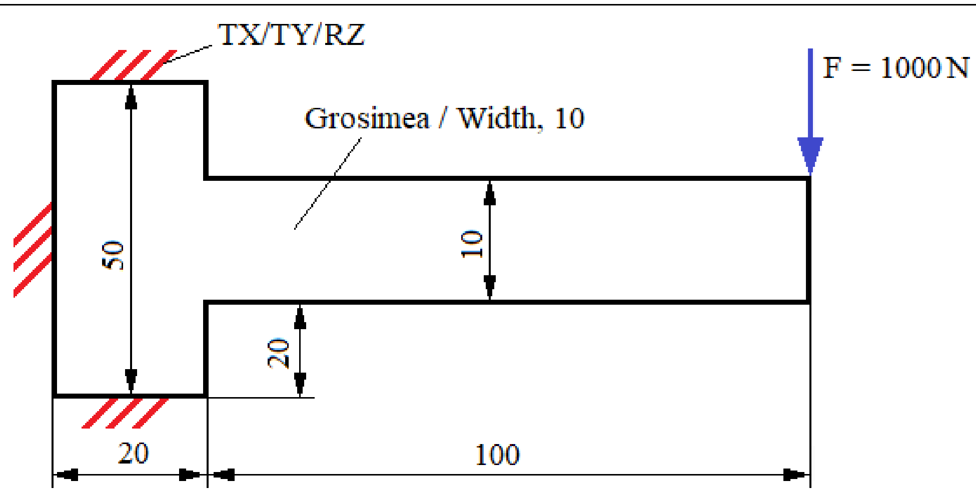
In the structure of the support device below, the elastic support element 1, firmly positioned in the body 2 through the bars 4 and the screws 5, must ensure a displacement imposed under the action of the press force F, developed by the slide 3, and return to the initial state after its cancellation.



A.3 The application goal
In this application, the analysis of the displacement, deformation and stress 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 stresss 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-A2 the effects of the singularities of concentration of stresss 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 behaviour
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-axes, 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 ↵ ✕].

Saving of the project

↵ Save As... → Save As, File name: [enter name, AEF-A.3] → ↵ 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, 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)

→ ↵ 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

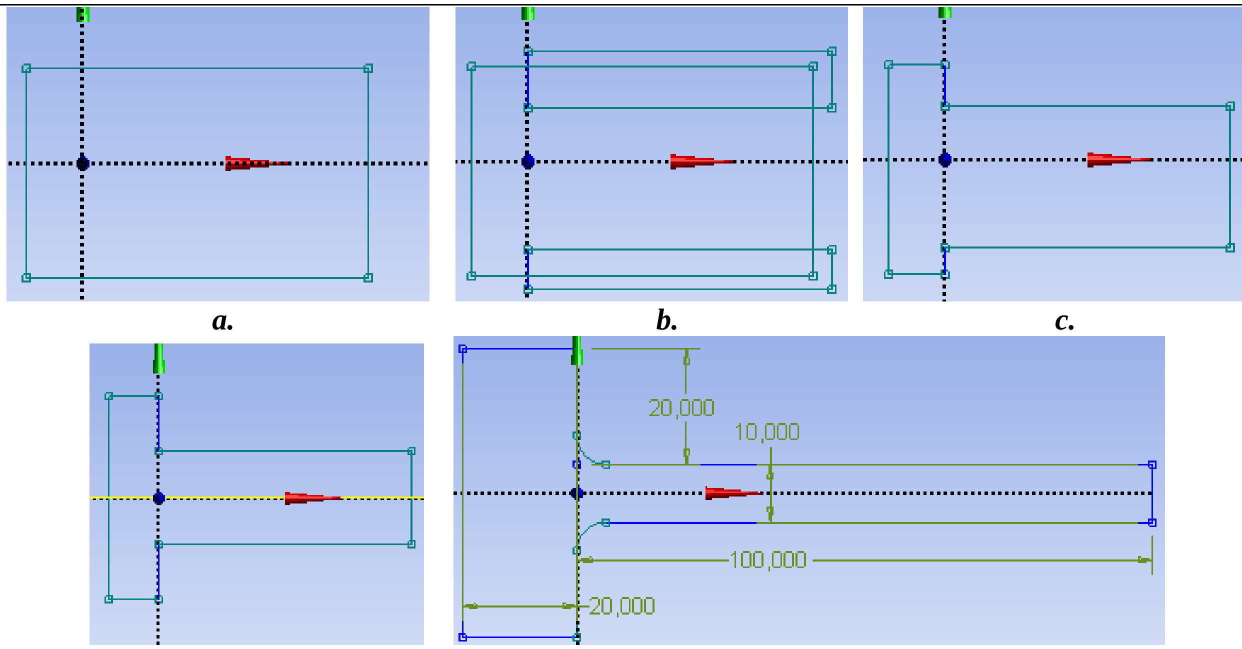
↵ Constraints →↵ Symmetry → [select with ↵ the X axis and then the two parallel lines with this axis to the left of the Y axis (fig. d)] → [select with ↵ the X axis and then the two parallel lines with this axis to the right of the Y axis (fig. d)].

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)]

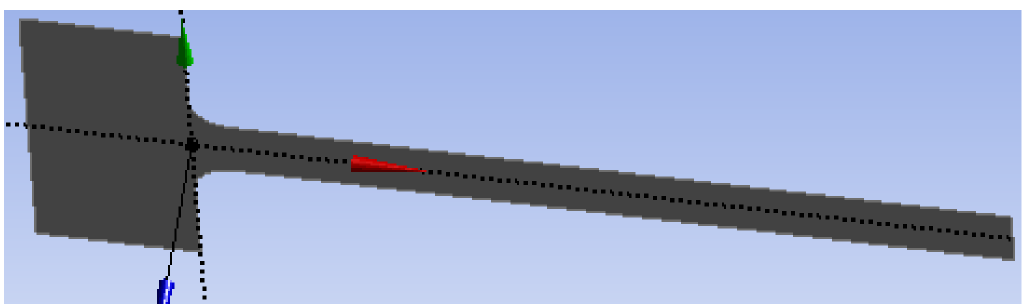


a. b. c.

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) ↵ ISO (axonometric visualization). ↵ ⊞ 1 Part, 1 Body → ↵ Surface Body → Details View, ⊟ Details of Surface Body : Body, [input name, *Suprafață bară*].



C.3.4 Saving 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

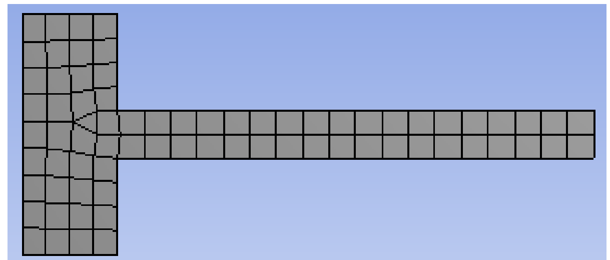
Outline → ↵ ⊞ Geometry → ↵ Suprafață bară → Details of "Suprafață bară": Material : Assignment, [is selected from the list ↵, ↵ Structural Steel] (usually, when there is only one material, this setting is default).

Setting the units

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

C.4.2 Model meshing

Case I (meshing with large first order finite elements)



Adopting the first order finite element (with the straight line, without intermediate node)

, Outline: → ↵ Mesh → Details of "Mesh", ⊞ Advanced : Element Midside Nodes, [select from list ↵, ↵ Dropped].

Automaticaly meshing

↳ Mesh → ↵ Generate Mesh.

Visualisation of meshing statistics

↵ Mesh → Details of "Mesh", ⊞ Statistics : ☐ Nodes, 106; ☐ Elements, 74.

Obs. It will be continued starting with step C.4.3 and after post-processing it will be returned and re-meshing according to the following case.

Case II (meshing with small first order finite elements]n singularities areas)

Adopting the first order finite element (with the second order line, with intermediate node)

↵ Mesh → Details of "Mesh", ⊞ Advanced : Element Midside Nodes, [selecting from the list ↵, ↵ Kept].

Setting global meshing

↵ Mesh → Details of "Mesh", ⊟ Defaults : ☐ Relevance, [modifying with ↵ valoarea / value, 100].

Setting local meshing to a point

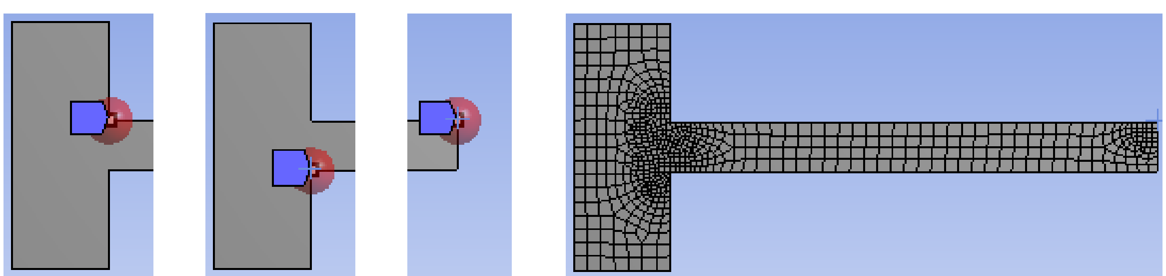
↳ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, ⊟ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ upper corner(fig. a)] → ↵ Apply; Sphere Radius, ↵Please Define → [input value 5]; Element Size, ↵Please Define → [input value, 1].

↳ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, ⊟ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ lower corner (fig. b)] → ↵ Apply; Sphere Radius, ↵Please Define → [input value, 5]; Element Size, ↵Please Define → [input value, 1].

↳ Mesh → ↵ Insert → ↵ Sizing → Details of "Sizing" - Sizing, ⊟ Scope : Geometry → ↵ (activating point selection filter) → [selecting with ↵ point of application of the force (fig. c)] → ↵ Apply; Sphere Radius, ↵Please Define → [input value, 5]; Element Size, ↵Please Define → [input value, 1].

Reviewing of meshing statistics

: Update → Outline: ↵ Mesh → Details of "Mesh", ⊞ Statistics : ☐ Nodes, 1954; ☐ Elements, 601.

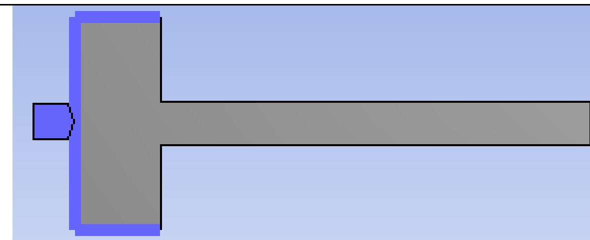
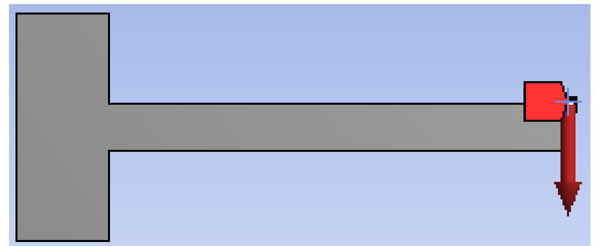


a. b. c. d.

Obs. It will be continued from step C.4.5

C.4.3 Supports and restraints modelling (fixed support)

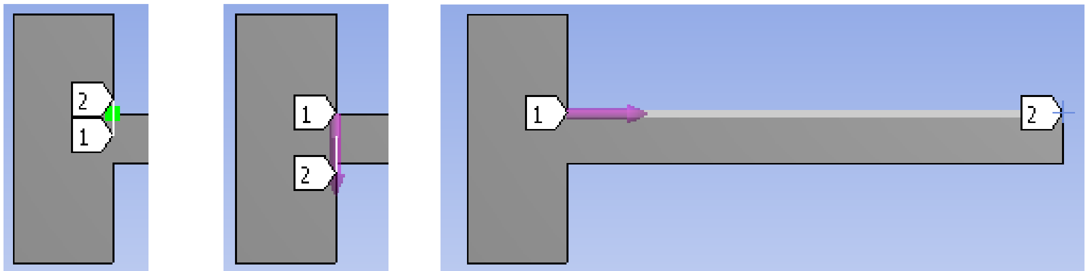
, Outline: ↵ ⊟ Static Structural (A5) → ↵ Supports → ↵ Fixed Support → Details of "Fixed Support"

, Geometry → ↵ (activating line selection filter) → [selecting with Ctrl+↵ fixed 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, [select from 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 Setting the results
Selecting the total displacements   , Outline: ↵ Solution (A6) → ↵ Deformation → ↵ Total.   *Selecting the normal stress on X axis*   ↵ Solution (A6) → ↵ Stress → ↵ Normal → Details of "Normal Stress", Definition: Orientation, [select from list ↵, ↵X Axis] (default).   *Selecting the tangential stress*   ↵ Solution (A6) → ↵ Stress → ↵ Shear.   *Selecting the equivalent stress*   ↵ Solution (A6) → ↵ Stress → ↵ Equivalent (von-Mises).   *Selecting the structural error*   ↵ Solution (A6) → ↵ Stress → ↵ Error.   *Selecting the normal stress on upper edge*   *Line generation:* Model (A4) → ↵ Construction Geometry → ↵ Path → Details of "Path", Start: ↵ Location, → ↵ (activating point selection filter) → [selecting with ↵ upper corner(fig. a)] → ↵ Apply (fig. b); End, ↵Location, ↵Click to Change → ↵ (activating point selection filter) → [select upper right peak (fig. c)] → ↵ Apply.   *Setting the normal stress on generated line*   ↵ Solution (A6) → ↵ Stress → ↵ Normal → ↵ Normal Stress 2 → Details of "Normal Stress 2", Scope: ↵Scoping Method, [selecting from list with ↵, ↵Path]; ↵Path, [selecting from list ↵, ↵Path].

Setting the equivalent stress on generated line: ↵ Solution (A6) → ↵ Stress → ↵ Equivalent (von-Mises) → Details of "Equivalent Stress 2" → ↵ Scope: ↵ Scoping Method, [selecting from list ↵ ▼, ↵ Path]; Path, [selecting from list with ↵ ▼, ↵ Path].



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

D.2 Launching the solving module
M: ↵ Solution (A6) → ↵ Solve.

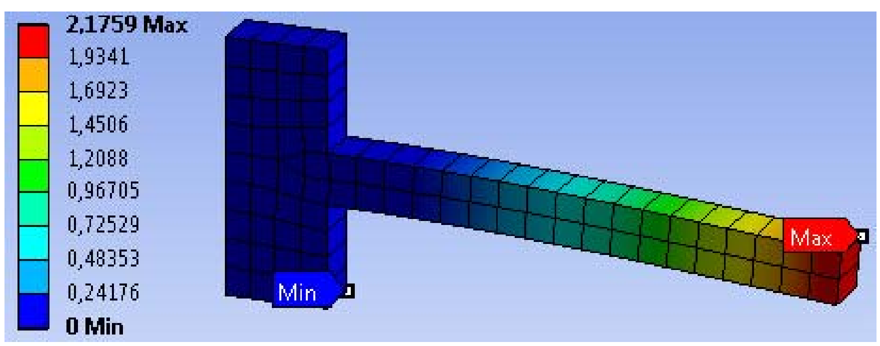
E. POST-PROCESSING OF RESULTS

E.1. Viewing the total displacement field

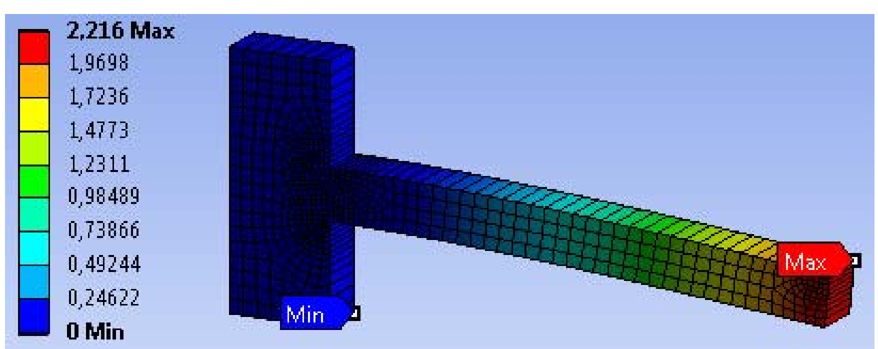
M, Outline: → ↵ Solution (A6) → ↵ Total Deformation; ↵ ISO (axonometric visualization); ↵ → [selecting from the list ↵ ▼, ↵ Smooth Contours] (visualization of smooth contours) ; ↵ → [selecting from the list ↵ ▼, ↵ Show Elements] (visualization of finite elements) ; ↵ Result → [selecting from the list ↵ ▼, ↵ 3, (Auto Scale)]; ↵ MAX (marking the node with the maximum total displacement); ↵ MIN (marking the node with the minimum total displacement).

Graph → ↵ Animation ▶ ■ (visualisation af animation).

Case I



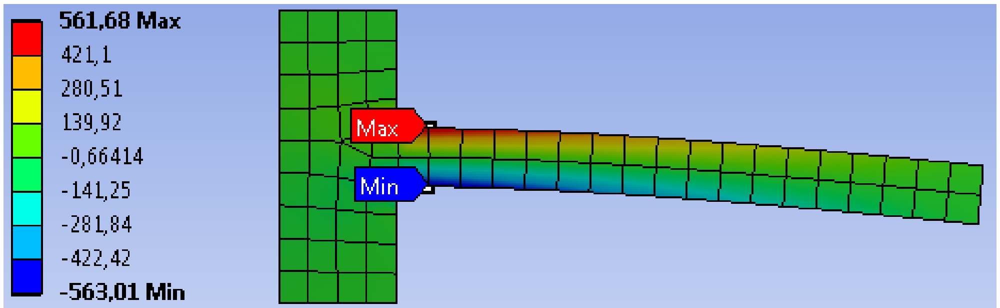
Case II

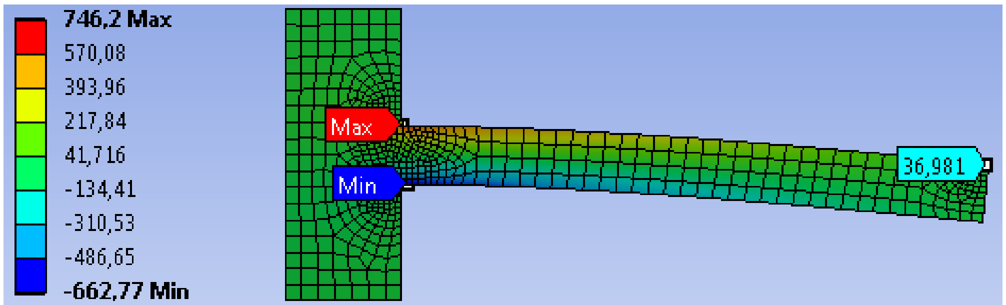


E.2. *Viewing stress field*
E.2.1 **Viewing the displacement field normal at the X axis**

Viewing the global field

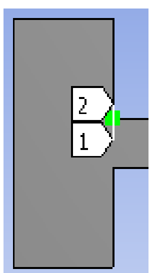
Outline: → ↵ Solution (A6) → ↵ Normal Stress; [selecting with ↳ grafical model] → ↵ View → Front.

Case I


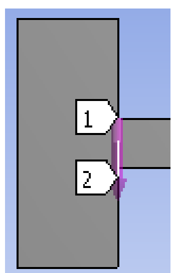
Case II


Viewing the field on a line

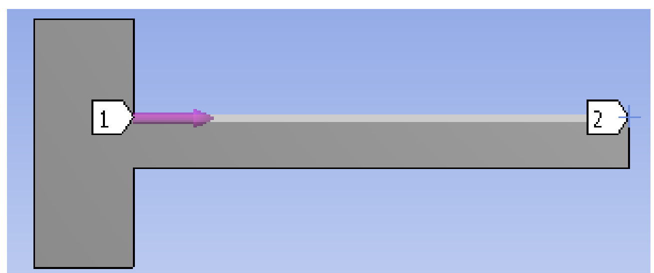
↳ Normal Stress 2 → ↵ Evaluate All Results (fig. d,e).



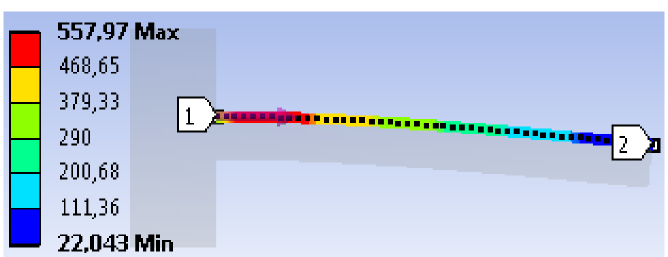
a.

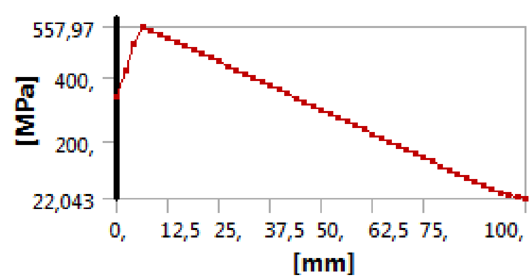


b.

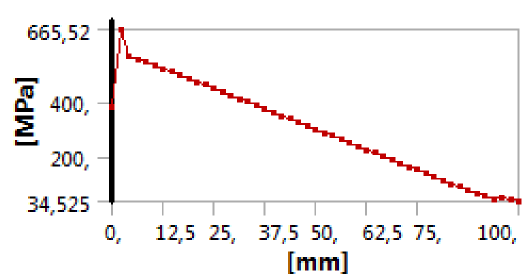


c.

Case I




Case II

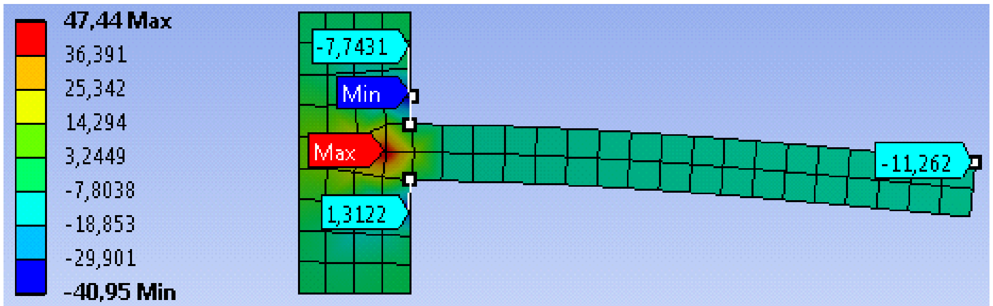



d. ***e.***

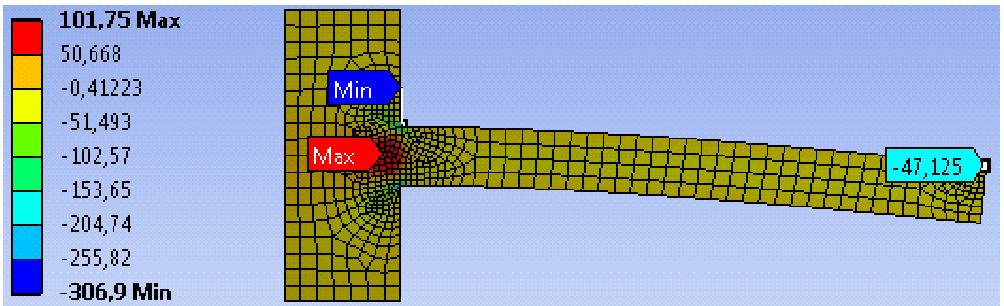
E.2.2 Viewing the tangential stress field

Outline: → ↵ Solution (A6) → ↵ Shear Stress.

Case I



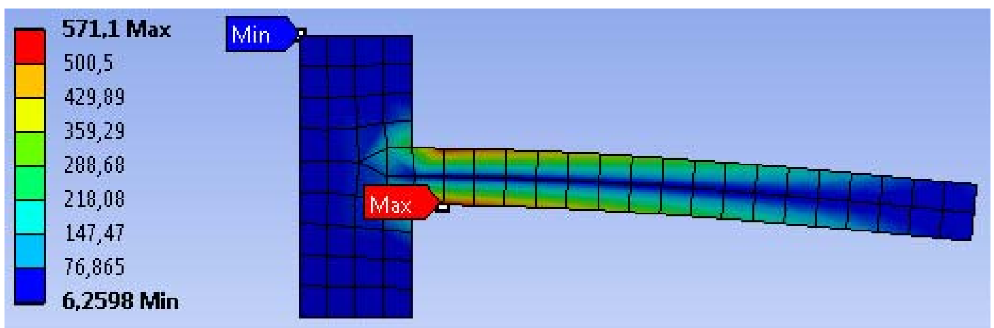
Case II



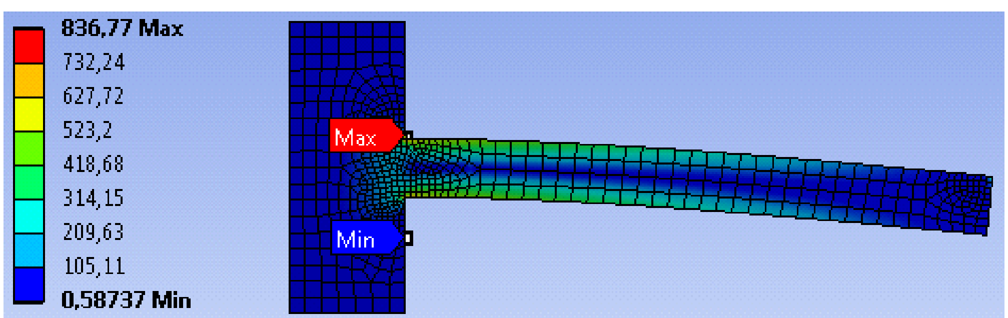
E.2.3 View the equivalent stress field (von Mises)

Outline: → ↵ Solution (A6) → ↵ Equivalent Stress.

Case I

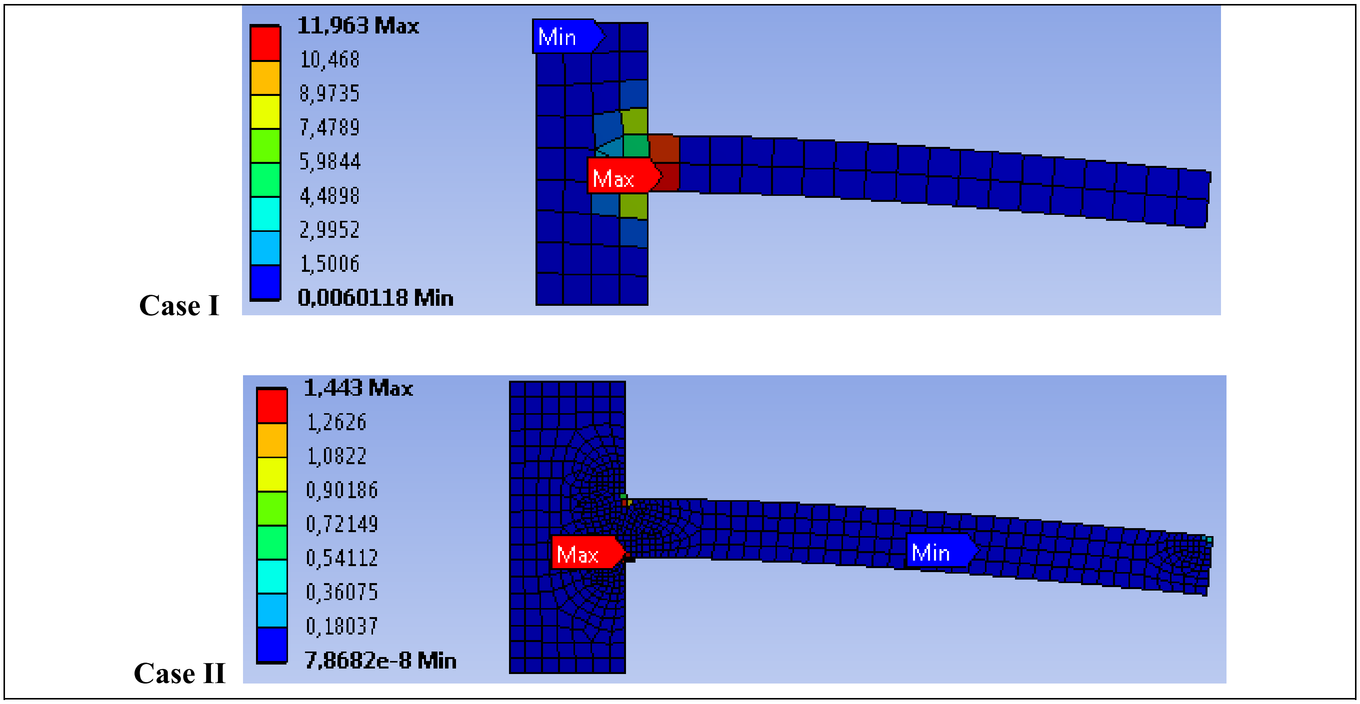


Case II



E.2.4 Viewing structural error fields

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



F. RESULTS ANALYSIS

F.1 The theoretical (analytical) calculation model

The classical analytical studies on the analysis structure (cantilever beam) are synthesized in the calculation of the following parameters (see application of AEF-A.1 subchapter F.1): maximum displacement, $\delta = 2$ mm, maximum normal bending tension (according to Navier's relation) , $\sigma_{î} = 600$ MPa, the maximum shear tangential stress (according to Juravschi's relation), $\tau_f = 15$ MPa.

F.2 Comparison and evaluation of results

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 (subcap. E and F.1) obtained under the conditions of the materials strength hypotheses, the following are highlighted:

- The maximum total displacement, 2,179 mm (case I) or 2,216 mm (case II), obtained with AEF (E.1), is almost equal to the displacement (2 mm) obtained from the theoretical analytical model (subchapter F.1).
- The maximum normal stress in the X direction, -563.2 MPa (case I) or 746.2 MPa, obtained by finite element analysis (subchapter E.2.2) has a deviation of -6% (case I) or 24.36 % MPa (case II) against the maximum normal stress (600 MPa) theoretical (subchapter F.1).
- The shear stress distribution (E.2.3) shows maximum values, 47.44 MPa (case I) or 101.75 MPa (case II), in the recessed area is 3.12 times (case I) or 6.78 or (case II) against the theoretical value, 15 MPa.
- The equivalent stress (von Mises) has the maximum value, 571.1 MPa (case I) or 836.72 MPa (case II) in the compressed and stretched area, respectively; it is observed that with the increase of the meshing fineness (case II) the value of the equivalent stress (von Mises) deviates by 39.4% due to the corner singularity (connection with null radius).

F.3 Accuracy analysis based on structural error

In subchapter. E.2.3 the structural error with the maximum value of 11.96 mJ (case I) or 1,443 mJ (case II) is highlighted; the maximum value in case I shows maximum errors of the stress in the fixing area.

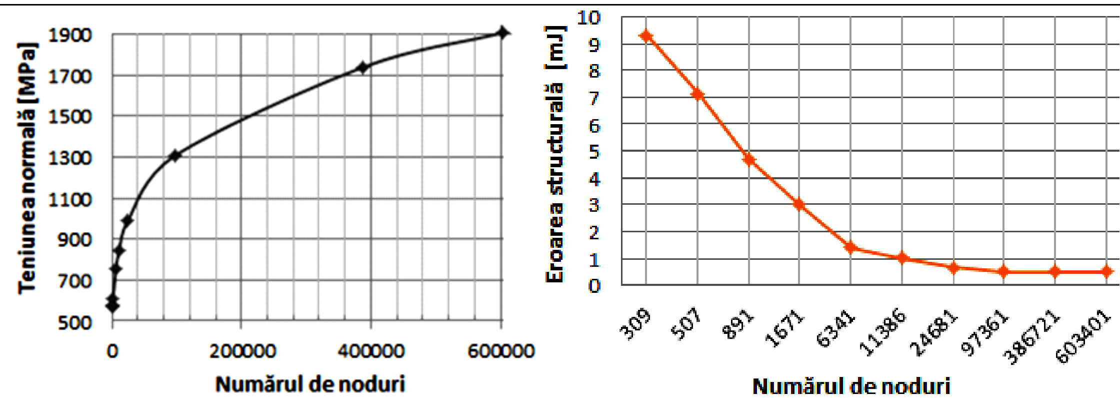
The structural error is determined as the difference of the deformation energies calculated using the average stresses associated with the finite element and the nodal stresses. The fineness of increased discretization leads to reduced structural error values and, therefore, it can be used on the one hand, as a global indicator of the discretization fineness, in the rediscretion of the entire structure and, on the other, as a local indicator of meshing fineness at local rediscretion.

In order to assess the accuracy of the stress type results, the field of structural error is analyzed, following a uniform distribution with reduced (preferably subunit) values of the structural error for acceptable accuracies; the areas where the structural error is increased in order to increase the precision of the results (the decrease of the structural error) will be made local rediscretion (subchapter E.2.3).

F.4 Analysis of convergence on X axis

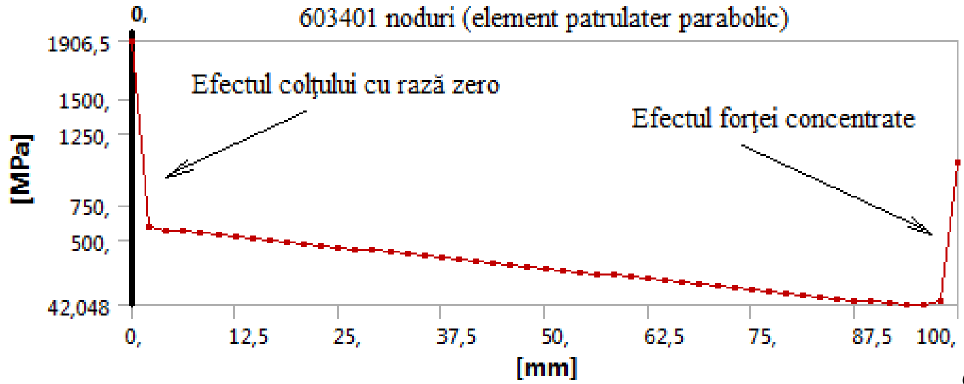
In order to highlight the effects of corner singularities (zero radius fillet) and concentrated force (point action), the model will be analyzed with various meshings, following the values of normal stress in the X direction, especially in the areas with singularities. For this purpose, the succession of modifying the fineness of meshing at the global level will be followed (the second order finite element set above will be kept):
↵ Mesh → Details of "Mesh", ⊟ Sizing : Use Advanced Size Function, [se selectează din listă cu / selecting from the list with ↵▾, ↵Off];
↵☐ Element Size, [se introduce valoarea dimensiunii elementului finit conform coloanei întâi din tabelul de mai jos / input the finite element size value according to the first column in the table below]. ↵ Mesh → ↵ Generate Mesh. ↵ Mesh → Details of "Mesh", ⊞ Statistics: [se evidenţiază numărul de noduri din caseta / showing nodes number, ☐ Nodes (coloana a treia / third column) şi numărul de elemente number of elements, ☐ Elements (coloana a doua / second column)].

Dimension EF [mm]	Number of EF	Number of nodes	/Normal stress [MPa]	Structural error [mJ]
5	80	309	572,96	9,3176
4	140	50	570,96	7,1435
3	258	891	577,32	4,712
2	500	1671	605,4	2,99
1	2000	6341	757,07	1,389
0,75	3200	11386	847,47	1,0124
0,5	8000	24681	990,35	0,651
0,25	32000	97361	1306,7	0,5
0,125	128000	386721	1737,1	0,5
0,1	200000	603401	1906,5	0,5



a. *b.*

At the corner point with singularity of the normal stress, its values increase with the increase of the number of nodes (there is no asymptote to tend to). The structural error decreases with the increase of the discretization fineness but at higher values of the nodes it has low values and it remains quasi-constant and the values of the normal stress increase non-asymptotically, which demonstrates the inconsistency of the process in the corner area with singularity (fig. A, c). The same situation specific to the singularity of the stress is observed in the area of application of the concentrated force (fig. C)



c.

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 stresss, for discretizations with increased fineness, shows that in the area with t singularities, although the structural error decreases to allowable values which would show a good accuracy, the values of the stresss do not converge towards the cvasiexact value, but they grow non-asymptotically.

The AEF model studied in this paper is inefficient in terms of modeling possibilities offered by the ANSYS platform because the connection area in the recess is null radius (theoretical case) and the force is concentrated at one point (also theoretical case). . These aspects are avoided in the application of AEF-A.2.