Application: AEF-A.1
Cantilever beam

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

CONTENT

A. PROBLEM DESCRIPTION

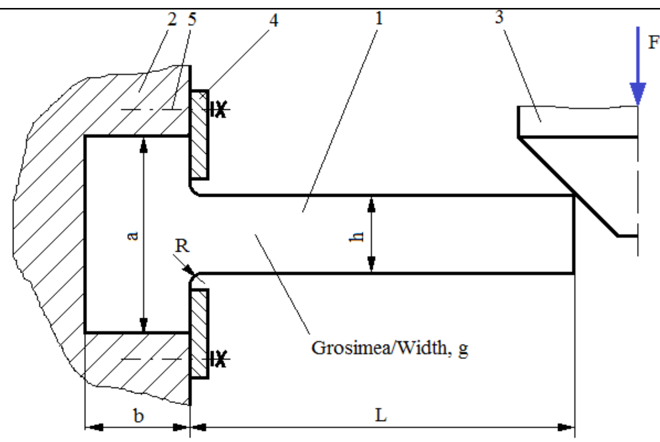
A.1 Introduction

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.

The primary objective of this application involves developing an AEF for a rectangular beam bar structure and comparing the results with the classical analytical ones.

A.2 Application description

In the structure of the support device below, the elastic support member 1, firmly positioned in the body 2 by the rods 4 and the screws 5, must provide a displacement imposed by the push force F developed by the skate 3 and return to the state initially after its cancellation.



A.3 The application goal

For this application, it is necessary to analyse the displacement, deformation and tension fields of the bearing element 1 made of C55 steel and having the following dimensions: L = 100, h = 10 mm, g = 10 mm, a = 50 mm, b = 20 mm. After analysing the structure from the fact that the element 1 has a constant thickness and the force loading, F = 1000 N, it produces evenly the width, it is highlighted the framing of the problem in the flat tension state (the tensions are invariant to thickness).

B. THE FEA MODEL

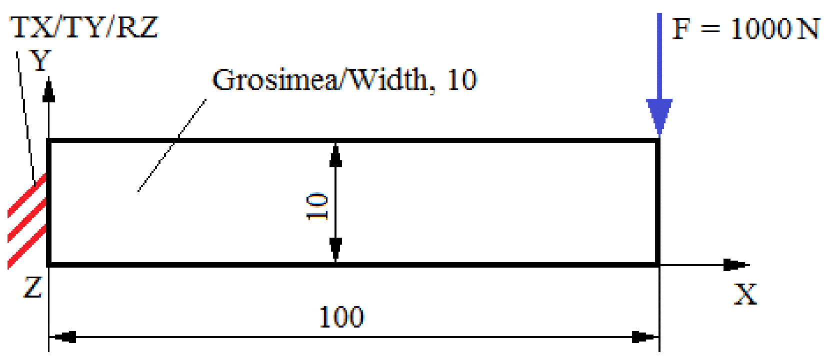
B.1 The model definition

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 is framed 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, ν = 0,3.

Average working temperature of the subassembly, T_0 = 20 ° C.

C. PREPROCESSING OF FEA MODEL

C.1 Creating, setting and saving the project

Creating of the project
Unsaved Project - Workbench, Toolbox: ↵ ⊟ Analysis Systems: ↵↵ Static Structural (the window with project modules appears automatically); [change name, Static Structural].
Setting of problem type (2D)
↳ Geometry → ↵ Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options, ↵ Analysis Type, [selecting from drop down list ↵▼, ↵ 2D] → [close the window, ↵ ✕].
Saving of the project
↵ Save As... → Save As, File name: [enter name, AEF-A.1] → ↵ Save.

C.2 Modelling of material and environment characteristics

→ Project Schematic → └ Engineering Data ✓ → ↲ Edit... → Outline of Schematic A2: Engineering Data: ↲ Structural Steel, Properties of Outline Row 3: Structural Steel: ⊟ Isotropic Elasticity → Young's Modulus, [selecting from drop down list, C (Unit) with ↲▼, ↲MPa], [enter in column, B (Unit) valoarea / value, 210000] → ↲ Update Project → ↲ Return to Project (others parameters are default).

C.3 Geometric modelling

C.3.1 Model loading, DesignModeler (DM)

→ Project Schematic → └ DM Geometry → ↲ DM New Geometry... → ANSYS Workbench: ↲ ◉ Millimeter, ↲ OK

C.3.2 Sketch generation

Viewing default plane (XY)

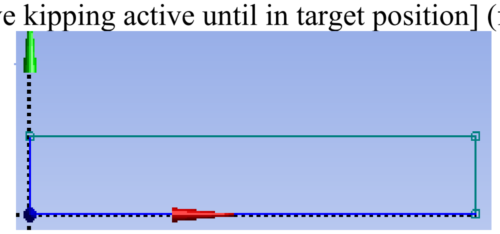
DM → ↲ Sketching → (Look At Face/Plane/Sketch) [automatically view of default plane, XY].

Rectangular line generation

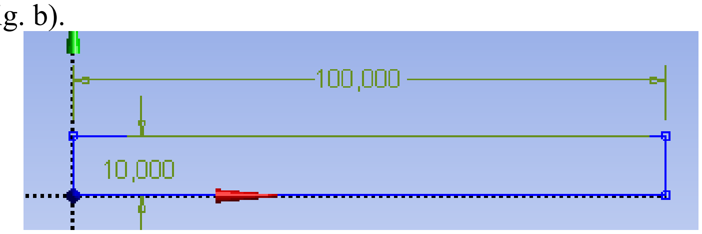
↲Draw → ↲ Rectangle → [trace rectangle line using pencil starting with, ↲, from coordinates system origin (appear symbol, P), and finish in opposite point simultaneously with release of the mouse fig. a)]

Dimensions

↲Dimensions → ↲ Semi-Automatic → [automatically create dimensions with ↲] → Details View, ⊟ Dimensions: 2: ☐ L1, [enter value, 10]; ☐ L2, [enter value, 100] (fig. b). ↲ Display (view dimenions), Name: ↲☐ (deactivate), Value: ↲☑ (activate). ↲ Move (move dimensions), [activate dimension with, ↲, and move kipping active until in target position] (fig. b).



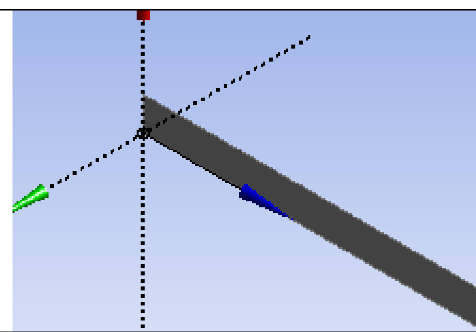
a.



b.

C.3.3 Surface generation

DM → ↲Concept → ↲ Surfaces From Sketches → Details View, ⊟ Details of SurfaceSk1: Base Objects → ↲ ⊞ XYPlane, ↲ ✓ Sketch1 → ↲ Apply; Thickness (>=0), [enter width value, 10] → ↲ Generate, └✓ Sketch1 → ↲ Hide Sketch (hide sketch). ↲ ISO (axonometric view).
↲ ⊞ 1 Part, 1 Body → ↲ ✓ Surface Body → Details View, ⊟ Details of Surface Body: Body, [enter name, *Suprafaţă bară*]



C.3.4 Save of geometric model

DM → ↲ (Save Project) → ↲ ✕ (Close).

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 → Project → ↲ ✓ Geometry → Details of "Geometry", ⊟ Definition: 2D Behavior, [selecting from drop down list ↲▼, ↲Plane Stress (default settings)].

Setting the material characteristics

Outline → ↲⊞ Geometry → ↲ Suprafață bară → Details of "Suprafață bară": Material : Assignment , [selecting from the list ↲▸, ↲Structural Steel usually, when there is only one item, this setting is default).

Setting the units

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

C.4.2 Model meshing

Automatic meshing (with implicit global parameters, including nonlinear finite element, parabolic with curved line side with an intermediate node)

M → Outline → Project → ↳ Mesh → ↲ Generate Mesh .

View mesh statistics

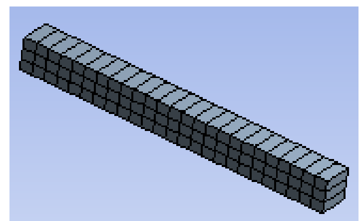
↲ Mesh → Details of "Mesh", ⊞ Statistics : Nodes , 282 ; Elements , 75 .

Adoption of the nonlinear finite element (with straight line or no intermediate node)

↲ Mesh → Details of "Mesh", ⊞ Advanced : Element Midside Nodes , [selecting from the list ↲▾, ↲Dropped] → Update .

Revision of mesh statistics

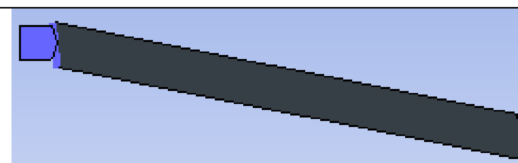
↲ Mesh → Details of "Mesh", ⊞ Statistics : Nodes , 104 ; Elements , 75 → ↲ISO .



Obs. The last linear finite element mesh has the same number of finite elements (75) but has a smaller number of nodes (104) than the number of nodes (282) corresponding to the parabolic finite element mesh.

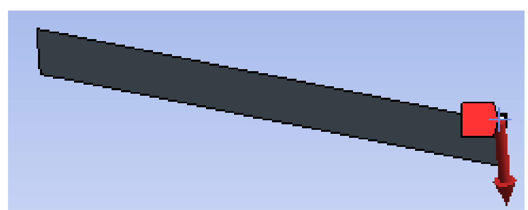
C.4.3 Supports and restraints modelling

M → Outline → ↲⊟ Static Structural (A5) → ↲ Supports → ↲ Fixed Support → Details of "Fixed Support", Geometry → ↲ (the line selection filter is activated) → [selecting with ↲ left edge] → ↲ Apply .



C.4.4 Loads modelling

M → Outline → ↲⊟ Static Structural (A5) → ↲ Loads → ↲ Force → Details of "Force", ⊟ Scope : Geometry → ↲ (the point selection filter is activated) → [selecting with ↲ the peak] → ↲ Apply ; ⊟ Definition : Define By , [selecting from drop down list ↲▾, ↲Components], Y Component , ↲ 0, N (ramped) → [input balue, -1000].



C.4.5 Saving the project

M → ↲ File → Save Project... .

D. SOLVING THE AEF MODEL

D.1. Selecting the results types

M → Outline → ↲⊟ Solution (A6) → ↲ Deformation → ↲ Total .

↲⊟ Solution (A6) → ↲ Stress → ↲ Normal → Details of "Normal Stress", ⊟ Definition : Orientation , [selecting from drop down list ↲▾, ↲X Axis] (default selection).

↵ Solution (A6) → ↵ Stress → ↵ Shear.

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

D.2 Launching the solving module

M → ↵ Solution (A6) → ↵ Solve.

E. POST-PROCESSING OF RESULTS

E.1. Viewing the displacement field

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

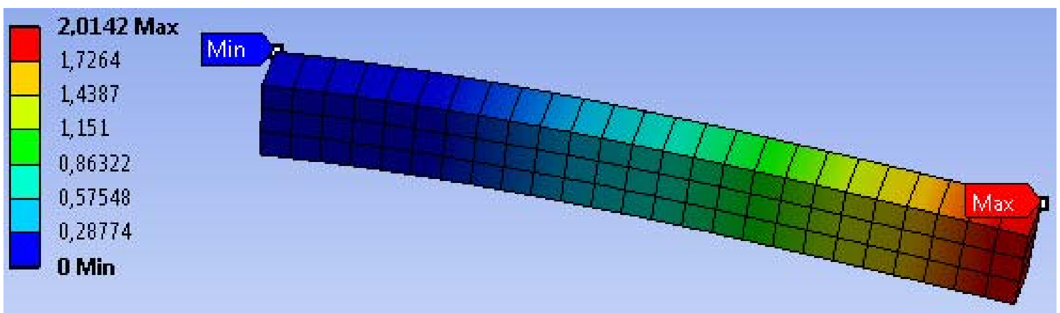
↵ → [selecting from drop down list ↵, ↵ Smooth Contours].

↵ → [selecting from drop down list ↵, ↵ Show Elements].

↵ Result → [selecting from drop down list ↵, ↵ 5, (2x Auto)].

Graph → ↵ Animation.

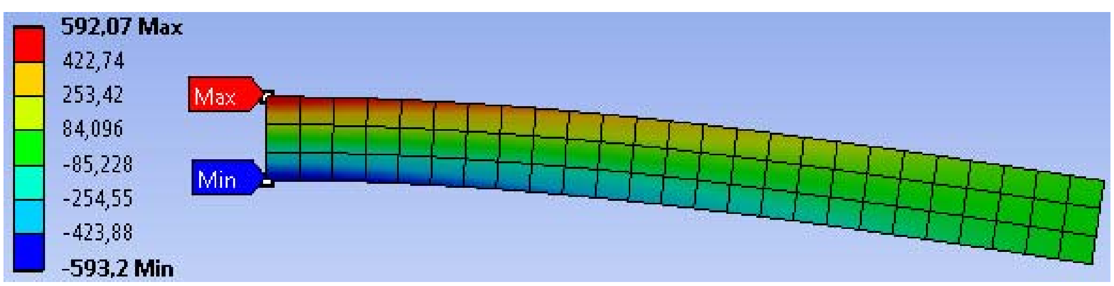
M → ↵ MAX; ↵ MIN.



E.2. Viewing the stress fields

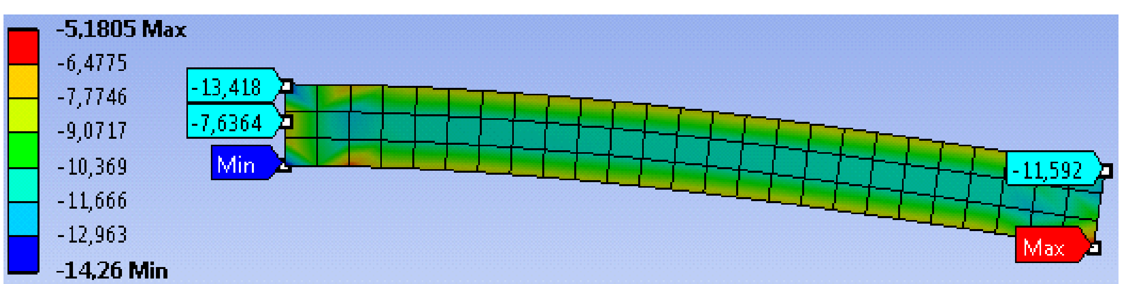
E.2.1 Viewing the normal stress in X direction

M → Outline → ↵ Solution (A6) → ↵ Normal Stress.

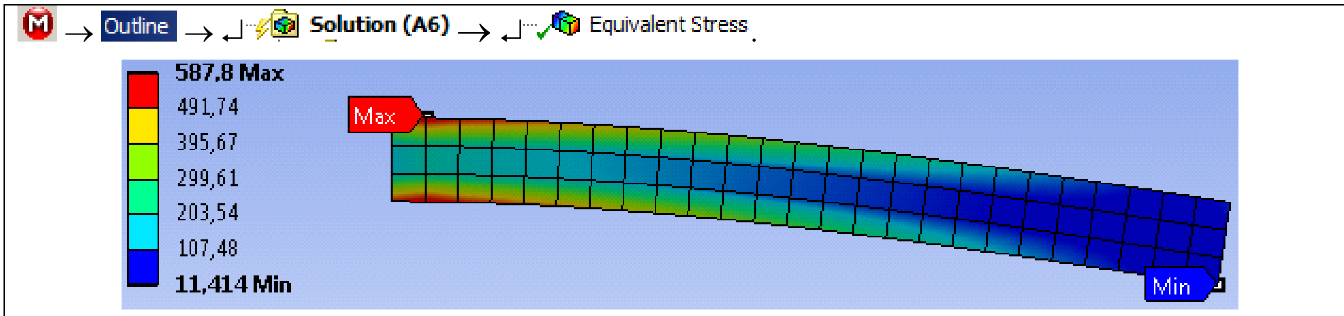


E.2.2 Viewing the tangential stress

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



E.2.3 Viewing the equivalent stress (von Mises)



F. RESULTS ANALYSIS

F.1 Theoretical (analytical) calculus model

Classical analytical studies on the analysed structure (embedded bar) can be synthesized in the calculation of the parameters:

- maximum displacement,

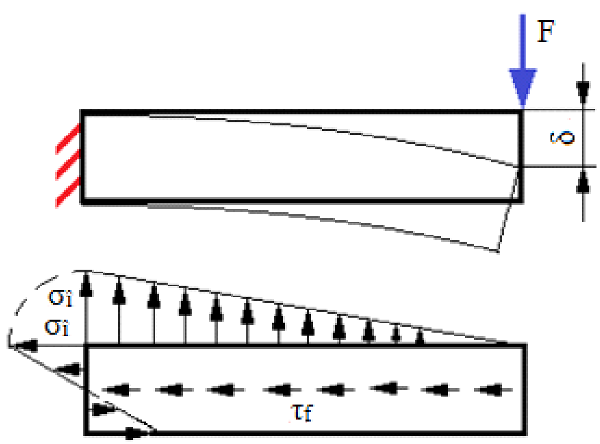
$$\delta = \frac{Fl^3}{3\,EI_z} = \frac{4\,Fl^3}{E\,b\,h^3} = \frac{4\;10^3\,10^6}{2\;10^5\;10\;10^3} = 2\;\text{mm};$$

- the maximum bending stress (Navier's relationship),

$$\sigma_i = \frac{M_i}{W_z} = \frac{6\,F\,l}{b\,h^2} = \frac{6\,10^3\,10^2}{10\,10^2} = 600\;\text{MPa};$$

- maximum tangential shear stress (Juravschi's relationship),

$$\tau_f = \frac{3}{2}\frac{T}{A} = \frac{3\,F}{2\,b\,h} = \frac{3\,10^3}{2\,10\,10} = 15\;\text{MPa}.$$



F.2 Comparing and evaluation of the results

Taking into consideration the results obtained using the modelling and AEF (sub-chapter E) and the use of the classical calculus relations (subchapters E and F.1) obtained under the conditions of the strengthens of the materials, the following are highlighted:

- the maximum displacement of 2,0142 mm obtained with AEF (E.1) is the same with the displacement (2 mm) obtained from the theoretical analytical model (sub-chapter F.1);
- the maximum normal stress in the X-direction, 593,2 MPa, obtained with the finite element analysis (sub-chapter E.2.2) has a -1,13% deviation from the theoretical maximum normal stress (600 MPa) (sub-chapter F. 1);
- the shear stress distribution (E.2.3) highlights maximum values (14.26 MPa) in the compressed clamp

area having a deviation of - 4.9% from the theoretical value (15 MPa);
- the equivalent stress (von Mises) has the maximum value (587.8 MPa) in the stretched clamp area.

G. CONCLUSIONS

On the first hand, the modelling and the finite element analysis from this application was done more with a teaching goal, in order to initiate the user with the main steps of developing an AEF application in ANSYS Workbench and, on the other hand, to compare and evaluate the results with some quasi-readings obtained through classical analytical models.

This process is recommended to be repeated for other practical situations in order to gain experience in developing analysis methods as well as evaluating the results.

The AEF model developed in this paper is inefficient from the point of view of the modelling possibilities offered by the ANSYS platform because it does not take into account the embedded connection area as well as the singularity associated with the concentric force due to the rough meshing with linear finite elements. These aspects are taken into account and studied in the application no. AEF-A.3.