Application: AEF-A.14
Static analysis of beam structures

KEY WORDS
Linear static analysis, Linear material, 1D geometric model, 1D finite element, Linear finite element, Beam structures, Lattice beams, Comparison with classical methods

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

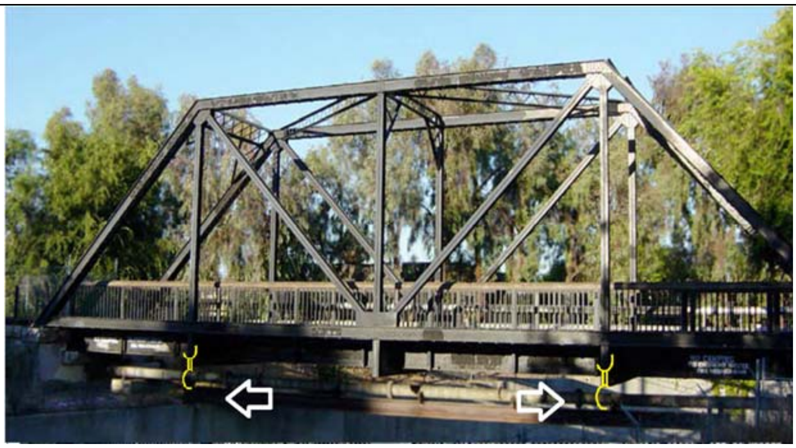
A.1 Introduction

The beams are primary semi-finished products with one of the dimensions much larger than the other two have various constant sections (circular, annular, square, rectangular, profiles, etc.).

The structures made of beams are specific, especially, to metal constructions (bridges, beams, pillars, trusses, etc.). For finite element analysis, beam structures are modeled with one - dimensional finite elements whose properties are determined by dimensional and orientation sectional parameters. These models substantially reduce the memory requirement as well as the computation time. The results obtained from these finite element analyzes are less valid in the nodal connecting areas (welds, riveted joints, screw assemblies) which can be analyzed separately using 3D and connecting finite elements.

A.2 Application description

In order to support a water supply pipe when crossing a river, it is necessary to create a beam-type structure with lattice. The pipe is attached to the supports on the beam, placed at equal intervals, using clamps. In order to avoid the occurrence of thermomechanical stresses at temperature variations, the beam is fixed at one end by means of a bolt assembly that allows rotation and at the other end it is supported and guided allowing translation.



A.3 Application goal
In the case of this application, the analysis of the fields of displacements, deformations and stresses of a statically stressed beam structure is presented in order to optimize its construction, respectively to minimize its weight in compliance with the deformation and strength restrictions. For the beginning of the finite element analysis, the supporting structure in the figure above is considered to be made of square S235 steel pipe with dimensions 80 x 80 x 5 mm. The dimensions of the support structure are: length L = 16a = 8 m and height H = 3a = 1.5 m. It is considered that this structure supports, in the lower part, a pipe that is suspended by means of two flanges, at equal distances margins (l = 2 m). For finite element analysis, the action of the supported pipe on the structure can be modeled by inserting in each node in which the pipe is attached by means of clamps a fixed force F = 5 KN. In addition, the consideration of internal forces of the own weight type is of particular importance for the analysis of these structures.

B. PREPARATION OF THE MODEL FOR ANALYSIS

B.1 The model definition

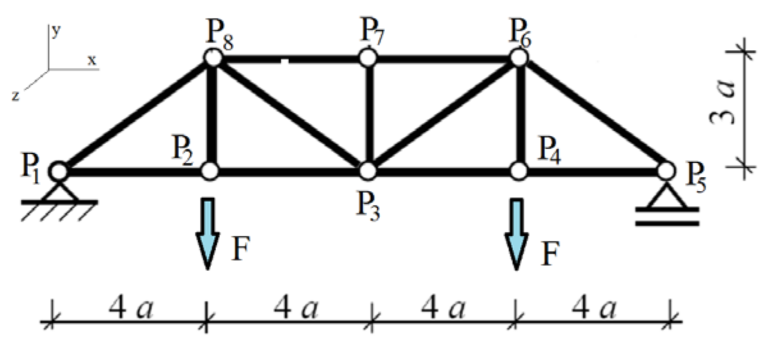
In order to draw up the finite element analysis model associated with the above application, it is necessary to identify:

- geometric shape and dimensions,
- restrictions induced by links with adjacent elements,
- external and internal loads (own weight),
- material characteristics.

B.2 The analysis model description

The geometric shape and the dimensions of the analysis model of the supporting structure are identical to those of the structure at the level of the sections of the sections. For the analysis, the structure is modeled with 1D finite elements and, therefore, the geometric model has the configuration in the figure below, having a = 0.5 m.

In order for the analysis model to have the same behavior as the real model, it is necessary to associate limit conditions that imply the cancellation of the translational displacements in relation to the OX, OY and OZ axes and of the rotations in relation to the OX and OY axes, in point P1, respectively of the rotations OX, OY and of the translations along the axes OY and OZ, in point P5. The structure of the analysis model is loaded with concentrated force F = 5 kN at points P2 and P4.



B.3 Characteristics of the material

For finite element analysis the strength characteristics of the material, S235 steel (equivalent to OL 37) are:

- modulus of longitudinal elasticity, E = 204,000 N / mm^2;
- transverse contraction coefficient (Poisson), ν = 0.3.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

Creating of the project

Toolbox: ↵ ⊟ Analysis Systems → ↵↵ Static Structural (the subproject window appears automatically); → [it can change the name Static Structural în *Structuri beame*].

Problem type setting (3D)

A : ↳ Geometry → ↵ Properties → Properties of Schematic A3: Geometry, ⊟ Advanced Geometry Options: ↵ Analysis Type, [select from list ↵▼, ↵ 3D] → [close window ↵ ×].

Saving of the project

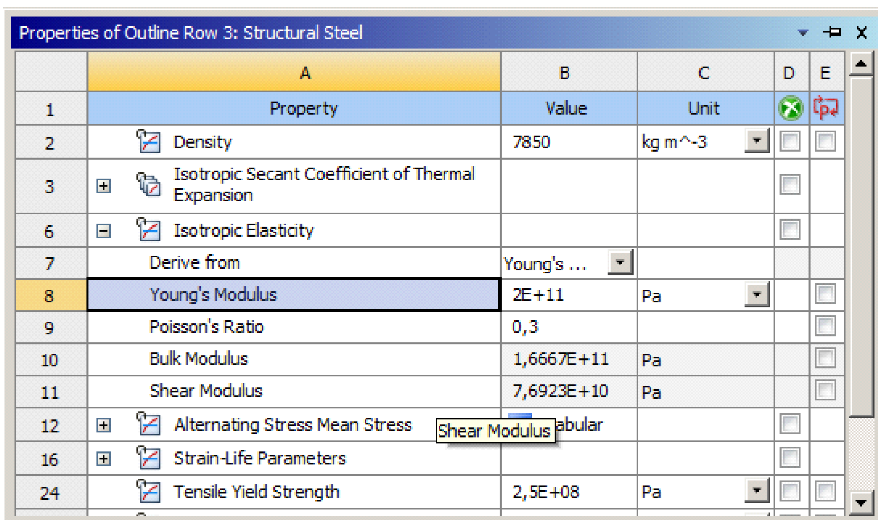
↵ Save As... → Save As, File name: [input name, *Structuri beame*] → ↵ Save.

C.2 Modelling of material 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, Young's Modulus, [select from list in column C (Unit) cu ↵▼, ↵MPa], [input in column B (Unit) valoarea / value, 204000] → ↵ Update Project → ↵ Return to Project (the other parameters remain the default).

If the window Properties of Outline Row 3: Structural Steel is not visible, the *Outline* and *Properties* or Reset Workspace options will be activated in the *View* menu.



Properties of Outline Row 3: Structural Steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m^-3		
3	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's ...			
8	Young's Modulus	2E+11	Pa		
9	Poisson's Ratio	0,3			
10	Bulk Modulus	1,6667E+11	Pa		
11	Shear Modulus	7,6923E+10	Pa		
12	Alternating Stress Mean Stress	Shear Modulus abular			
16	Strain-Life Parameters				
24	Tensile Yield Strength	2,5E+08	Pa		

C.3 *Creating the geometric model*

C.3.1 Uploading DesignModeler Module (DM)

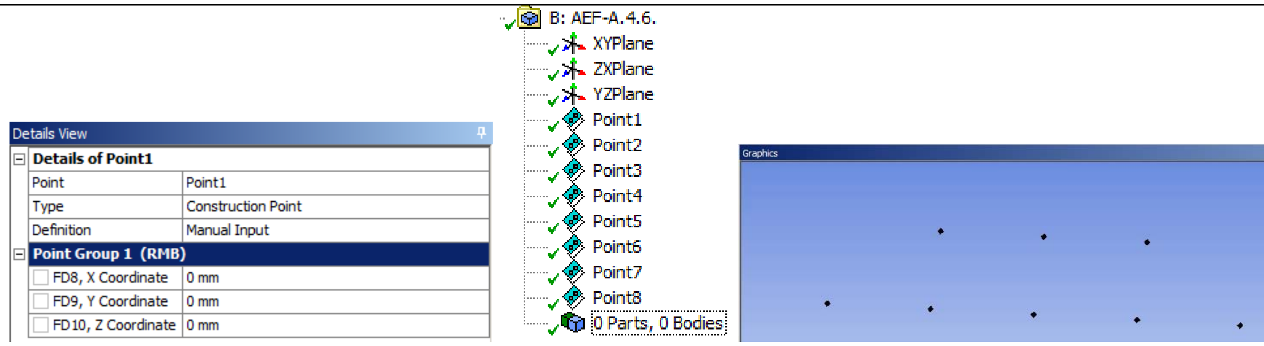
, Project Schematic: ↳ Geometry → ↵ DM New Geometry... → ANSYS Workbench: ↵ Millimeter, ↵OK.

C.3.2 Generating points

DM → Modeling → Create → Point [in the 3D modeling area the point P1 is created based on the Cartesian coordinates] → Details View → **Details of Point 1** → Definition ▼: Manual Input; **Point Group 1 (RMB)** → x = 0; y = 0; z = 0 → Generate.

The points P2… P8 are constructed in the same way, using the resulting Cartesian coordinates based on the dimensions given in the model for analysis:

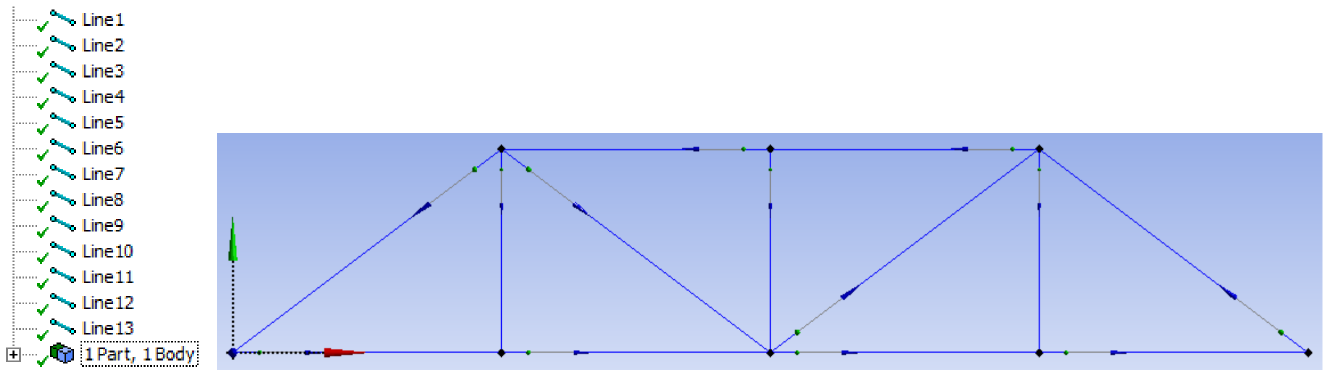
P2 (2000; 0); P3 (4000; 0); P4 (6000; 0); P5 (8000; 0); P6 (6000; 1500); P7 (4000; 1500); P8 (2000; 1500).



C.3.3 Generating of beam sections

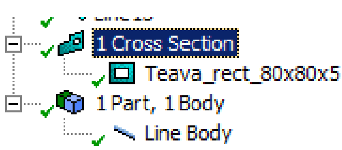
→ Modeling → Concept → Lines From Points → Details View → **Details of line 1** → Point Segments [while holding down the Ctrl key, select points P1 and P2]: Apply → Generate.

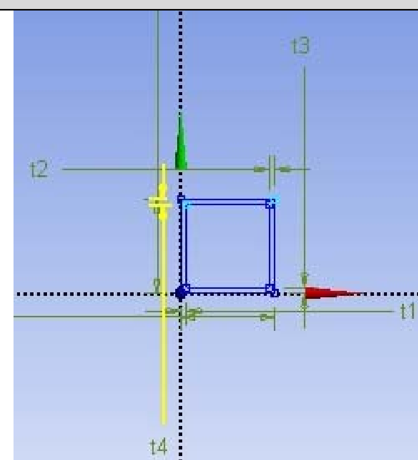
In the same way the segments corresponding to the pairs of points are constructed: (P2, P3), (P3, P4), (P4, P5), (P5, P6), (P6, P7), (P7, P8), (P8, P1), (P8, P2), (P8, P3), (P7, P3), (P6, P3), (P6, P4). The structure shown below will be obtained.



C.3.4 Segment section generation

→ Modeling → Concept → Cross Section → Rectangular Tube → Details View → **Details of Rect Tube 1** → Sketch: Teava_rect_80x80x5; **Dimensions:** W1 = 80 mm, W2 = 80 mm, t1 = 5 mm, t2 = 5 mm, t3 = 5 mm, t4 = 5mm → Generate.



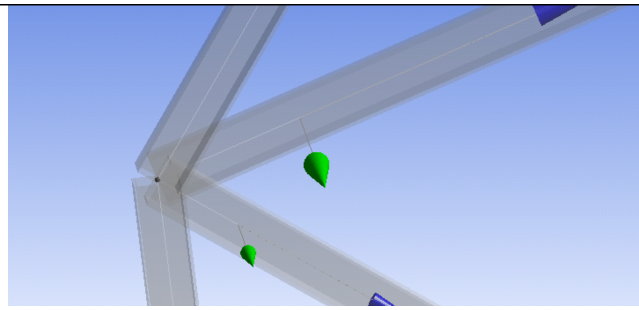


C.3.5 Assigning a transverse profile to the metal structure

Generating (building) a profile does not mean assigning it to a 1D beam structure. To complete the procedure, proceed as follows:

→ ↵ Line Body → Details View → **Details of Line Body** → Cross Section: Teava_rect_80x80x5;

Modeling → View → Cross Section Solids [the option of 3D visualization of the sections is activated] → [check that the profiles are oriented correctly; the profile orientation is viewed using the green section view option is activated by green arrows; if the profile is not symmetrical and is not project oriented, the orientation can be changed accordingly: ↵ Line Body → Details View → **Line Body-Edge** → Reverse Orientation?: Yes] → Offset Type: Centroid → Generate.



C.3.6 Saving the geometric model

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

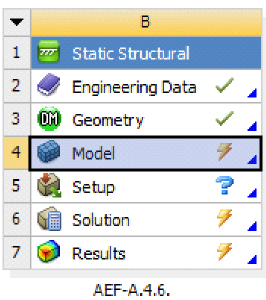
C.4 Finite element modelling

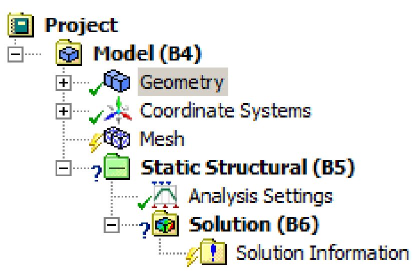
C.4.1 Launching the finite element modeling module and set the material characteristics and problem type

Launching of the finite element modeling module

, Project Schematic ↵↵ Model → launching module *Mechanical [ANSYS Multiphysics]*.

↵ Geometry → Details of "Geometry" → **Definition** → Element Control : Program Controlled .





Details of "Geometry"

Definition	
Source	C:_Documente\Carte MEF\AEF-A_Aplicatie ferma 1D\F...
Type	DesignModeler
Length Unit	Millimeters
Element Control	Program Controlled
Display Style	Body Color
Bounding Box	
Properties	
Statistics	
Basic Geometry Options	
Advanced Geometry Options	

Setting the unit of measurement 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 → ↵ Details of "Geometry" → **Material** : ↵ Assignment → [select from list ↵ , ↵ Structural Steel (default)].

Setting the model type

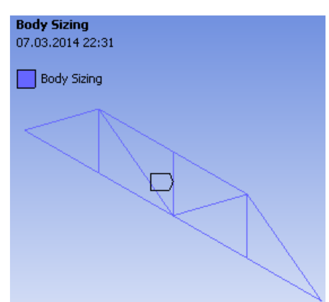
Outline: ↵ Geometry → Details of "Geometry", **Definition** : ↵ 3D Behavior.

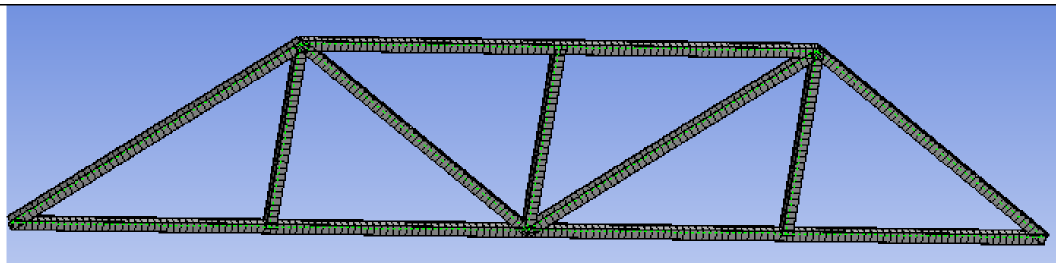
C.4.2 Model discretization and finite element size setting

, Outline: ↵ Mesh → Mesh Control → Sizing → Details of "Sizing" - Sizing → **Scope** → Select Geometry: [will be selected with ↵ structure geometry, using the selection filter (Body)] Apply; **Definition Element** → Size: 0,05 m → Update

Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	5,e-002 m
Behavior	Soft

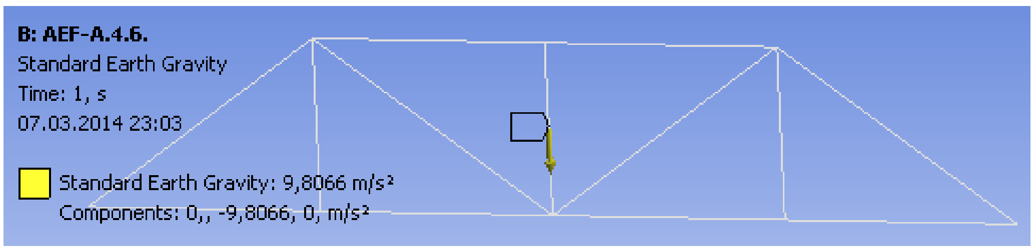




C.5 Supports and restraints modelling

Introduction of gravitational acceleration

M → ↲?Static Structural (B5) → Inertial → Standard Earth Gravity (the selection of the gravitational acceleration implies the taking into account of the own weight of the metallic structure) → Details of "Standard Earth Gravity" → **Definition** → Direction: -Y Direction.

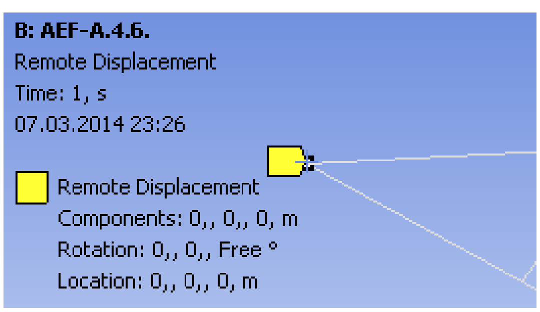


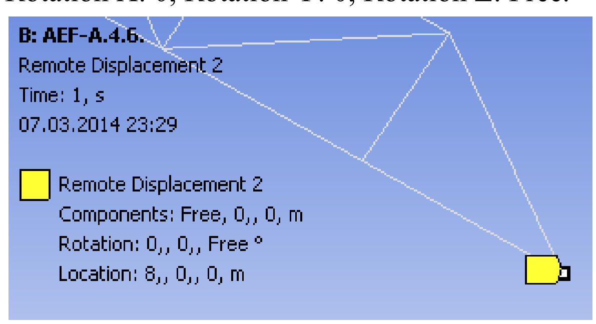
Input restraint

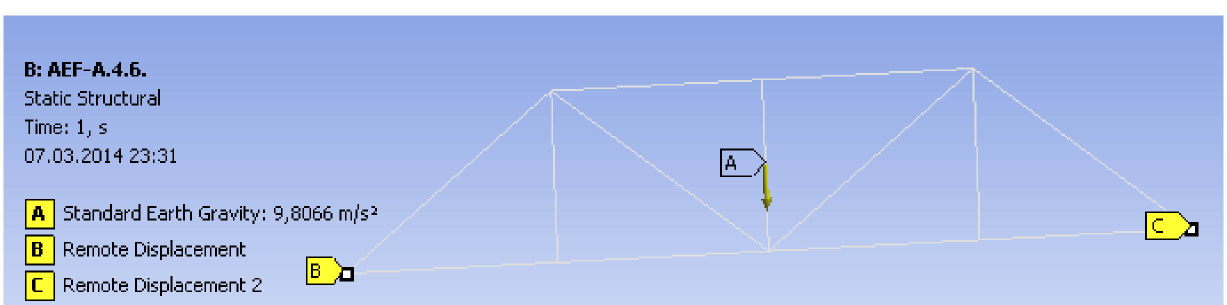
M → ↲?Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" → **Scope** → Geometry: [will be selected with ↲ point P1, using the option (Vertex)] → Apply; **Definition** → X Component: 0, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.

The procedure will be repeated for point P5:

M → ↲?Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" → **Scope** → Geometry: [will be selected with ↲ point P5, using the option (Vertex)] → Apply; **Definition** → X Component: Free, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.







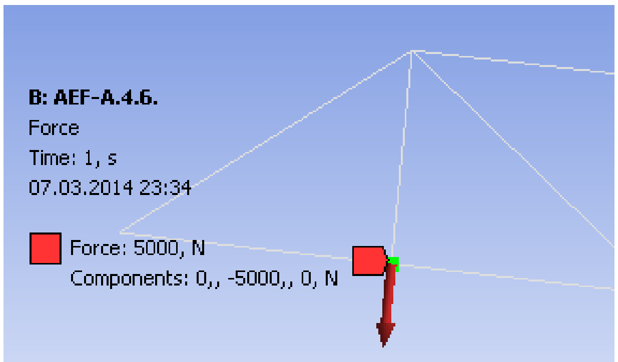
C.6 Load modeling

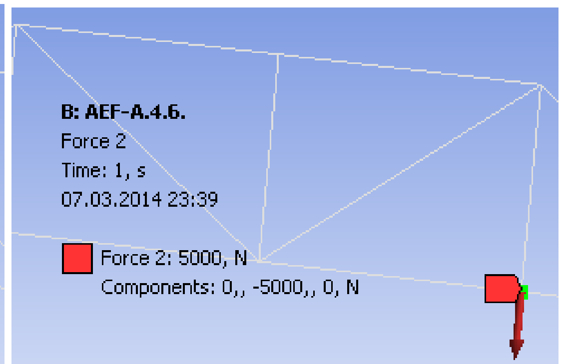
Input forces in selected point

→ ↲ Static Structural (B5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [select point P2 using option (Vertex)] → Apply; **Definition** → Magnitude: 5000 N; Direction: axa Y [a segment of the metal structure parallel to the OY axis will be selected with ↲].

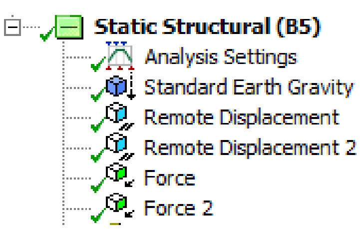
The procedure will be repeated for point P4:

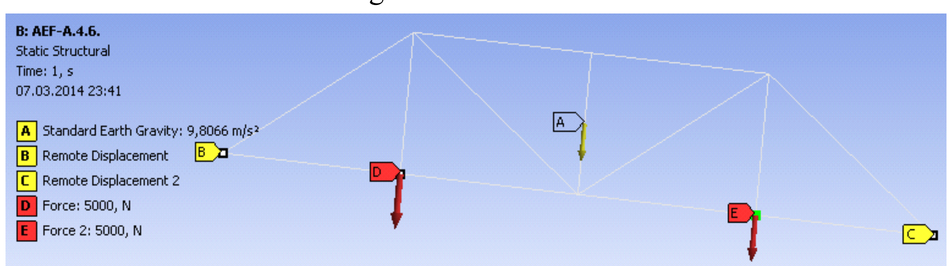
→ ↲ Static Structural (B5) → Loads → Force → Details of "Force" → **Scope** → Geometry: [select point P2 using option (Vertex)] → Apply; **Definition** → Magnitude: 5000 N; Direction: axa Y [a segment of the metal structure parallel to the OY axis will be selected with ↲].





The constraints and loads of the structure will look like the figure below.





D. SOLVING THE FEA MODEL

D.1 Launching the calculation module and select the types of results

In order to select the final data types to be analyzed after the launch of the calculation module, the series of commands presented below will be followed. → ↳ Solution (B6) → Insert → Deformation → Total [use the commands in the open command box with ↳].

Acelaşi rezultat se poate obţine prin utilizarea comenzilor:

↲ Solution (B6) → Deformation → Total [the buttons in the menu beams are used]

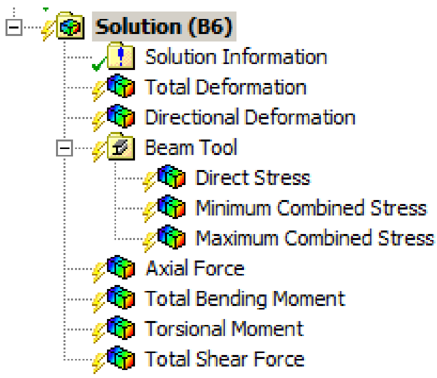
and

↲ Solution (B6) → Deformation → Directional.

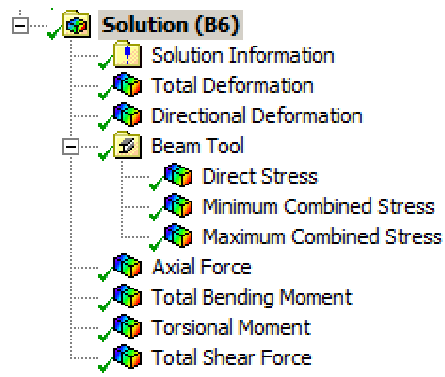
For this type of structure, the Beam tool can be applied in order to visualize the linearized stresses on the component elements. It is customary, in the process of designing beam structures, to take into account the components of axial stresses that arise from the effect of axial and bending loads in all directions. The following are the other types of results to be analyzed:

↲ Solution (B6) → Tools → Beam Tool.

↵ Solution (B6) → Beam Results → Axial Force.
↵ Solution (B6) → Beam Results → Bending Moment.
↵ Solution (B6) → Beam Results → Torsional Moment.
↵ Solution (B6) → Beam Results → Shear Force.
D.2. Launching the solving module
→ ↵ Solution (B6) → Solve.



→ Solve →



E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu beams:

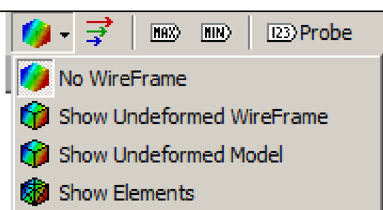
Result 8,6e+002 (Auto Scale) → Result 1.0 (True Scale).

Total deformation view

↵ Solution (A6) → Total Deformation → Graph → Animation.

If the images are not suggestive enough, in terms of how the work is distorted, you can return to changing the display scale by selecting a higher value: Result 1,7e+003 (2x Auto).

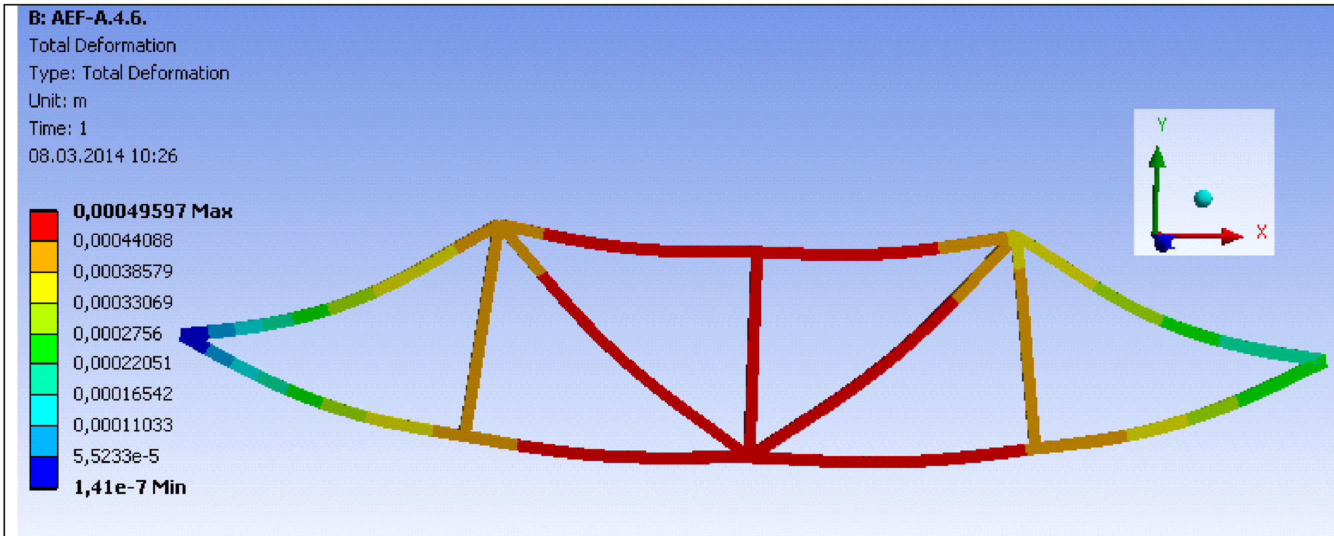
Various forms of distorted state representation can be used by calling the (Edge) button. Show Showformed WireFrame will be selected, an option that displays the undeformed and warped models in the same representation.



The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command.

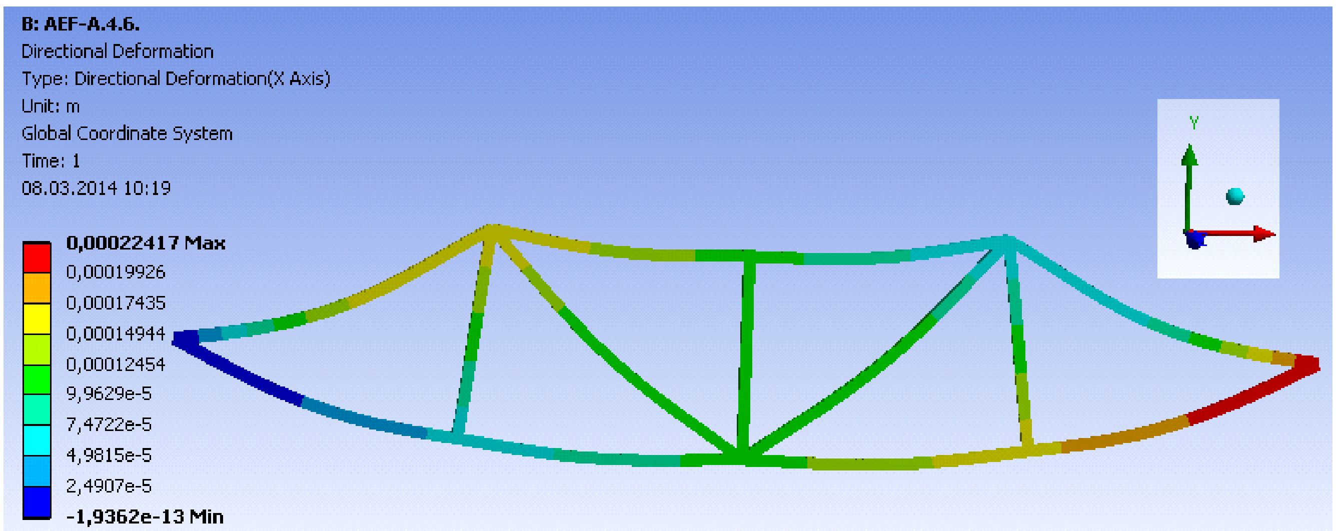
Vizualizare deformației totale / Total deformation view

↵ Solution (B6) → Total Deformation → Graph → Animation.



Visualization of the deformation in a certain direction

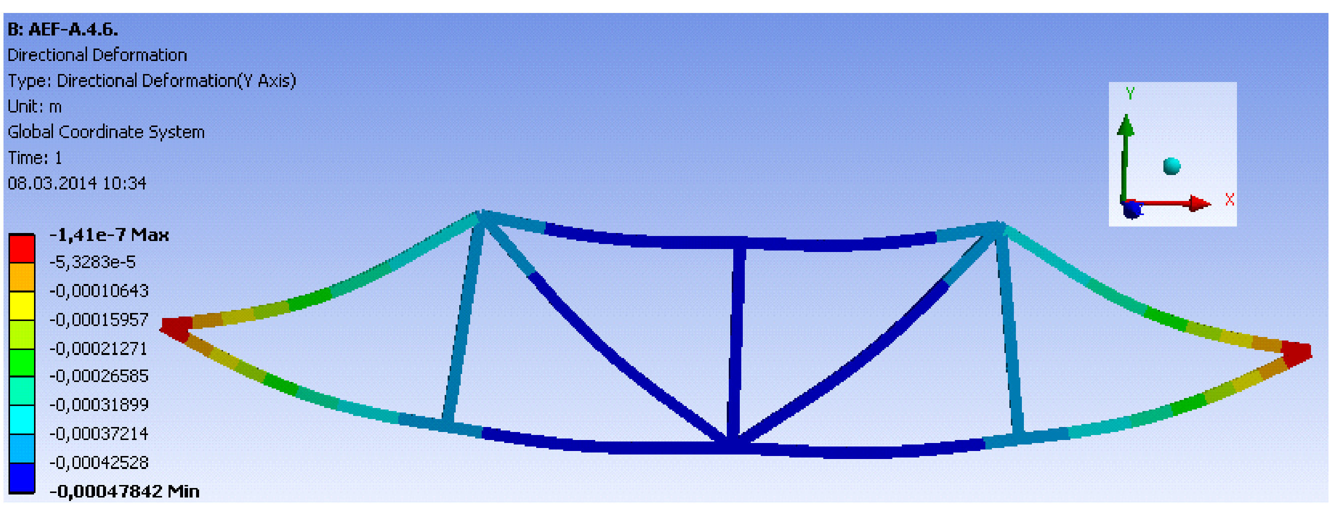
↵ Solution (B6) → Directional Deformation → Graph → Animation.



If you want to view in another direction, follow the steps below:

↵ Solution (B6) → Directional Deformation → Details of „Directional Deformation" → **Definition** →

Orientation : Y Axis → Solve.

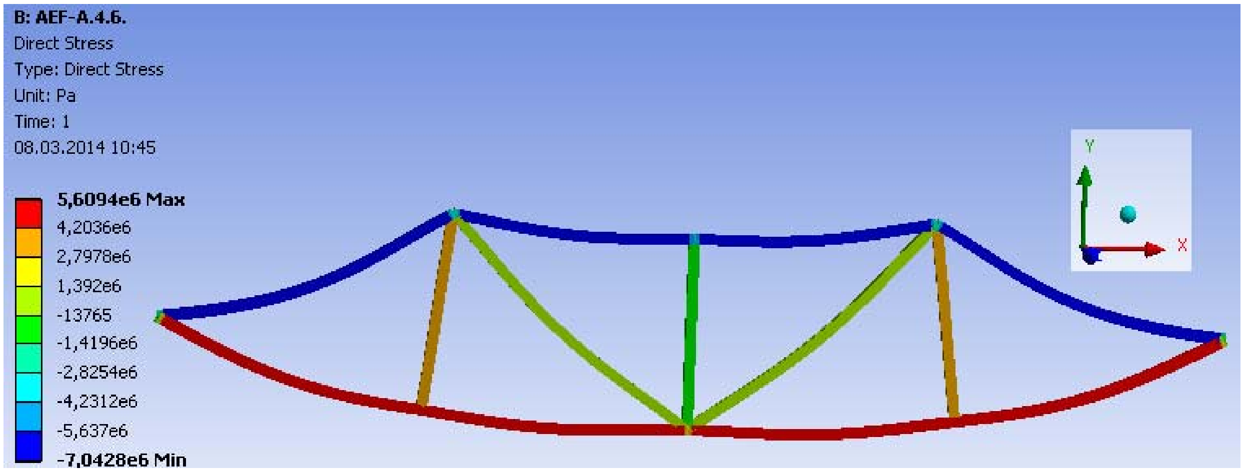


E.2. Visualizing the fields of stresses, forces and moments

<u>*Direct Stress*</u>

Direct Stress (σ_x) represents the component of the internal tension due to the axial force in a section of the beam.

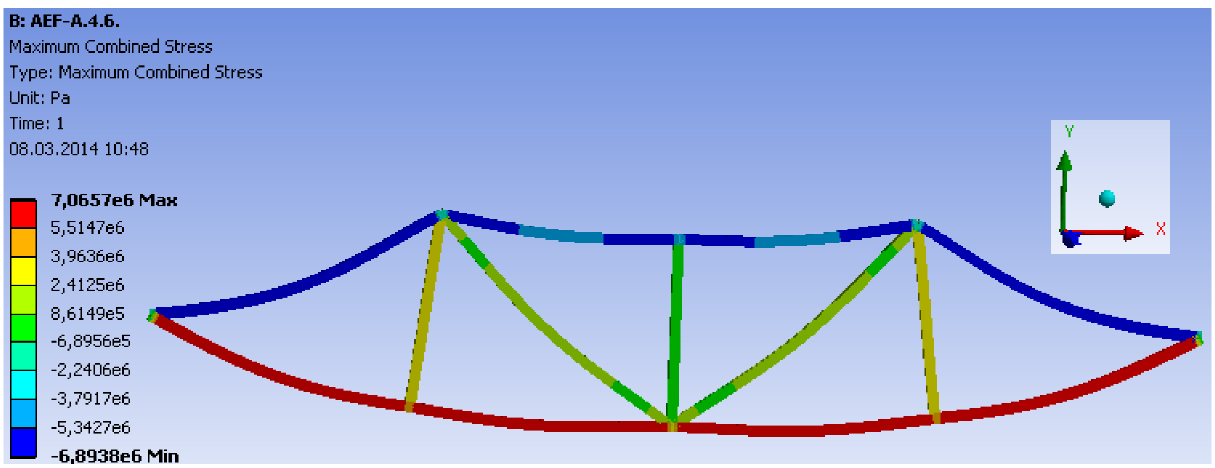
↵ Solution (B6) → Direct Stress → Graph → Animation.



<u>*Maximum Combined Stress*</u>

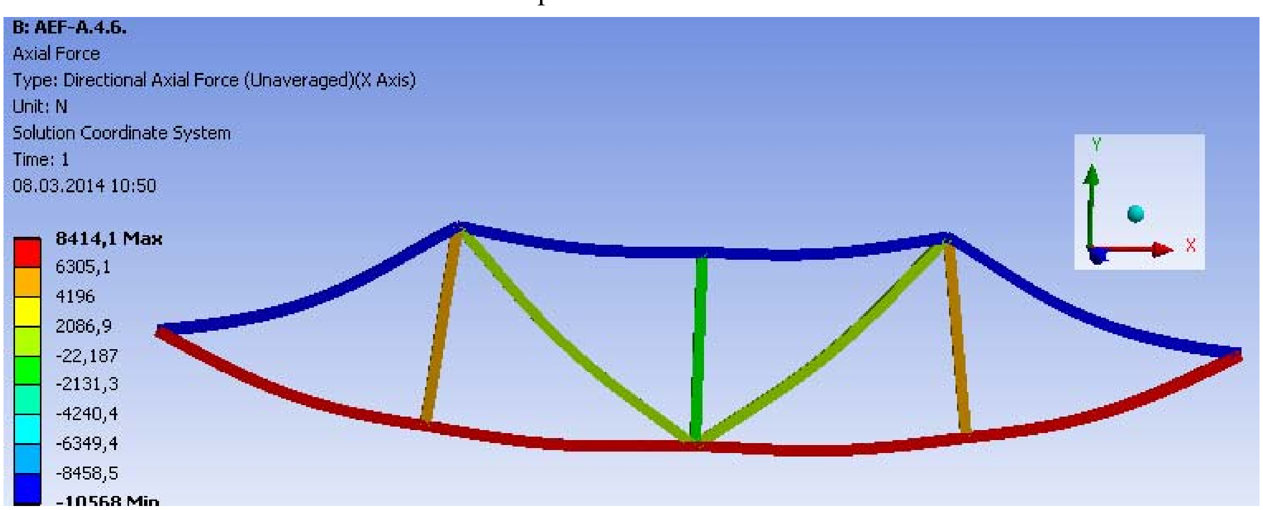
Maximum Combined Stress – represents a linear combination of *Direct Stress* şi / and *Maximum Bending Stress*.

↵ Solution (B6) → Maximum Combined Stress → Graph → Animation.



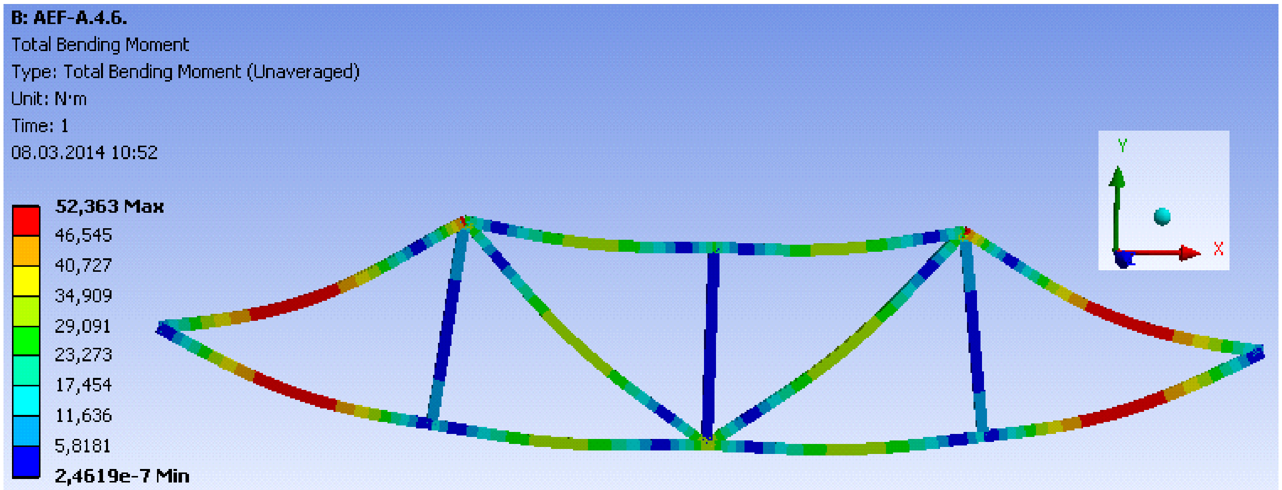
<u>*Directional Axial Force*</u>

↵ Solution (B6) → Axial Force → Graph → Animation.



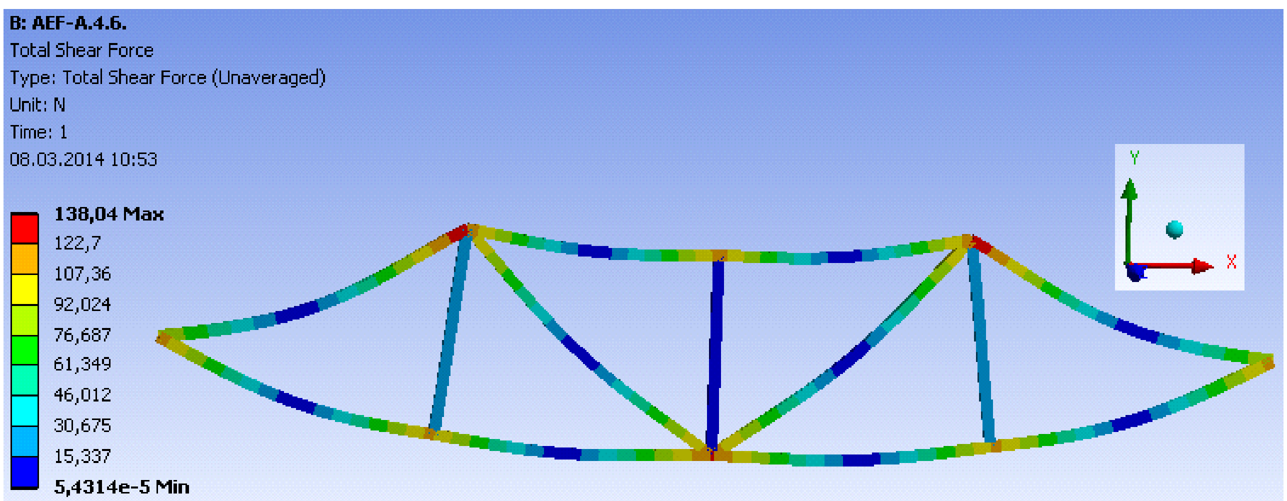
Bending Moment

↲ Solution (B6) → Total Bending Moment → Graph → Animation.



Total Shear Force

↲ Solution (B6) → Total Shear Force → Graph → Animation.



F. ANALYSIS OF RESULTS

F.1 / Analysis of the results obtained by FEM
It is observed that, despite the fact that the modeling of the beam structure was performed with the help of 1D bodies, the results obtained are suggestive, being presented in a 3D environment. From the point of view of the total deformations, it is observed that the maximum value is 0.5 mm in the middle area of the metal structure. In the Ox direction, the maximum displacement is obtained in the bearing corresponding to point P5, having a relatively small value, 0.2 mm. It is observed that the areas with high shear efforts are those corresponding to the assembly points of the sections and those required for bending being the middle areas of the sections (explained by the maximum value of the forces arms in the nodes). Examining the graphical representation of the axial forces, it is observed that the sections located in the lower part of the structure (segments 1-2, 2-3, 3-4, 4-5) are subject to stretching stress - represented in red and those located in the upper part of the structure (1-8, 8-7, 7-6, 6-5) are required for compression - represented by the color blue. The sections in the middle of the structure (8-3, 7-3, 6-3) are very little stressed axially, the value of the efforts tending towards 0.

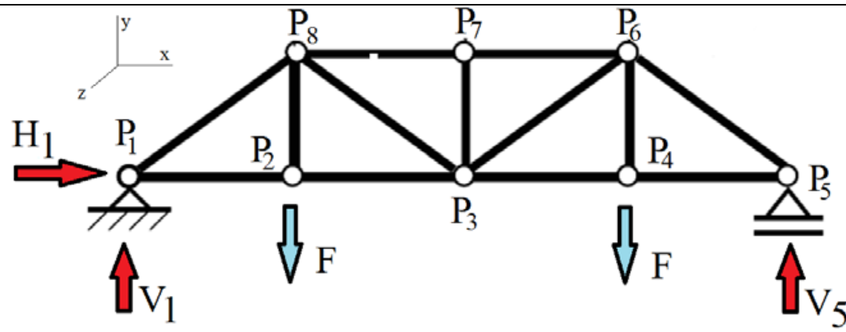
The information regarding the deformations, corroborated with the information regarding the internal stresses, the combined maximum stresses lead to the conclusion that the structure withstands loads without problems, the values of maximum stresses not exceeding 7 x 10^6 Pa, value well below the allowed material limit.

F.2 Prezentarea rezultatelor obţinute prin metoda clasică / Presentation of the results obtained by the classical method

Number the nodes: N = 8.

The number of beams b = 13 is established, as well as the number of simple external connections r = 3.

The static determination condition is checked: 2N = b + r.



Calculate the reactions in the bearings corresponding to points P1 and P5 using the solidification method, writing the equilibrium equations.

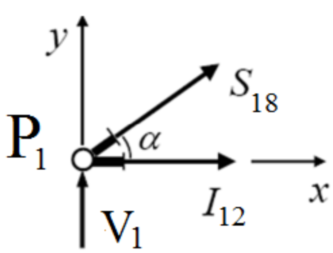
$$\sum M_{(P1)} = -F \cdot 4a - F \cdot 12a + V_5 \cdot 16a = 0$$

$$\sum M_{(P5)} = -F \cdot 4a - F \cdot 12a + V_1 \cdot 16a = 0$$

$$\sum F_{(Oy)} = V_1 + V_5 - 2F = 0$$

We obtain the results: $V_1 = V_5 = F = 5.000$ N; $H_1 = 0$.

Isolate node P1 and represent it graphically, writing the equilibrium equations.



$$\sum F_{(Ox)} = I_{12} + S_{18} cos\alpha = 0$$

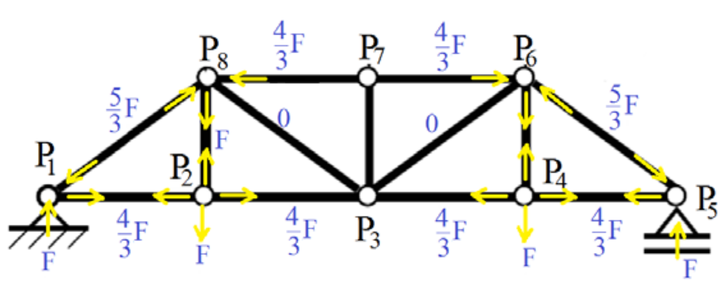
$$\sum F_{(Oy)} = F + S_{18} sin\alpha = 0$$

Given that, according to the imposed geometry of the beam structure, $\cos\alpha = 4/5$ and $\sin\alpha = 3/5$, the following values are obtained for the axial stresses in sections 1-2 and 1-8, depending on the force F :

$I_{12} = 4/3$ F = 6.666 N;

$S_{18} = -5/3$ F = -8.333 N

In the same way, the other axial stresses in each section of the beam structure will be calculated. The following distribution of efforts will be obtained:



F.3 Comparative analysis of results
Using classical methods of *Strength of Materials*, the results are obtained by relatively simple calculations but which require significant time resources, which are directly proportional to the level of complexity of the analyzed structure.

G. CONCLUSIONS

Several sub-chapters of the analysis can be addressed in this subchapter.

From the point of view of the pre-processing phase, it can be seen that the use of 1D bodies involves minimal resources for both modeling and discretization. Another strong point is that the transverse profile of the sections can be modified / oriented very easily, without influencing the basic shape of the beam structure. Moreover, it is possible to use different profiles for each section. The sections can be connected in several ways, depending on the central axis of the profiles used.

The introduction of supports, constraints and demands is quick and easy. The declaration of the materials, as well as the discretization of the beam structure are controllable processes, which can be done automatically or manually.

Comparing the results obtained by the classical method and FEM, it can be seen that they are comparable, at least in the case of axial stresses, a case that was calculated classically, the finite element method providing much more data, over time and with much resource consumption. smaller.

It can be seen that the structure of the beams is very little required, and much smaller profiles can be used in order to achieve savings. Changing the profile of the beam sections and recalculating is done in a very short time, being an easy procedure.

For example, the 80 x 80 x 5 mm rectangular pipe profile will be replaced with 50 x 50 x 5. The result of *Maximum Combined Stress* is shown below.

→ Modeling → Concept → Cross Section → Rectangular Tube → Details View → **Details of Rect Tube 2** → Sketch: Teava_rect_50x50x5; **Dimensions:** W1 = 50 mm, W2 = 50 mm, t1 = 5 mm, t2 = 5 mm, t3 = 5 mm, t4 = 5 mm → Generate.

→ ↲ Line Body → Details View → **Details of Line Body** → Cross Section : Teava_rect_50x50x5;

→ ↲ → Line Body → Update Selected Parts → Update: Use Geometry Parameter Value → Solve.

As the section decreases, the values of the combined stresses and strains increase, but do not reach the maximum permissible values.