Application: AEF-A.15
Self-induced vibration modes

KEY WORDS
Linear static analysis, modal analysis, eigenfrequencies, eigenmodes, eigenmodes, planar geometric model, linear material, 1D finite element, linear finite element

CONTENT

A. PROBLEM DESCRIPTION

A.1 Introduction

In many practical situations for the design of complex mechanical systems it is necessary to know the own frequencies and modes of vibration of some components or even of the whole. These parameters, invariable with time, determined in the conditions of observing the equilibrium configuration, are intimate characteristics of the analyzed structure depending on the shape, dimensions and material.

The determination of the frequencies and of the own vibration nodes of the mechanical components or structures can be done by means of the modal analysis. Natural frequencies and vibration modes are very important parameters for the design phase because they provide information about the dynamic behavior of the analyzed structures. The modal analysis within the ANSYS program is a linear analysis. Any nonlinearity such as plasticity and contact elements is ignored, even if it is defined. Modal analysis is used to calculate the natural frequencies and modes of deformation of the structure.

A.2 Application description

Bridges made of lattice beams are characterized by high rigidity, being generally made of elements made of steel. Identifying your own modes and frequencies of vibration is particularly important to take into account in the design process the values of the frequencies of certain demands: natural (earthquakes, strong winds) or artificial (induction of vibrations by vehicles crossing the bridge) to avoid total or partial destruction of the structure. At the same time, because tensions can occur in the structure at temperature variations, the bridge is fixed at one end by means of a rotating joint with a bolt and at the other end it is supported and guided allowing translation.

A.3 Application goal
The purpose of this application is to identify its own modes and frequencies of vibration for the bridge-type structure of lattice beams in order to avoid its resonance phenomena. The displacement fields for each vibration mode will be presented in order to optimize the construction of the structure, respectively to minimize its weight in compliance with the deformation and strength restrictions.

B. PREPARATION OF THE MODEL FOR ANALYSIS

B.1 The model definition

The modal solution is obtained following a modal analysis which consists in completing the following steps:

- model construction;
- applying loads and obtaining the solution through structural analysis;
- expanding modes;
- viewing the results.

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 basic equation solved in a typical unamortized modal analysis typical for the ANSYS program is given by the classical problem of eigenvalues:

$$[K] = \omega i^2 [M]$$

where [K] is the stiffness matrix; is the shape vector (eigenvector) of mode i;

ω_i is the natural frequency of mode i (ω_i^2 is the eigenvalue); [M] is the mass matrix.

Among the methods for solving this equation, recommended in the ANSYS program, the Lanczos vectors method will be used in this paper. The static stresses of the bridge-type mechanical structures can be overlapped by the dynamic stresses which, together with the static ones, can cause the destruction of this part. One of the dynamic stresses to which a bridge is subjected is the stress due to vibrations caused by various causes during use (passing of people, vehicles, vibrations due to machinery or work equipment, weather stresses - strong wind, etc.).

The mechanical structure studied in this paper is considered independent, without mechanical connections and other constraints. This method of calculation was approached because the modeling of related elements would lead to large dimensions of finite element models, which would have a negative effect on the accuracy of the results. Thus, the modal analysis of the mechanical structure will be performed in order to obtain indications on the occurrence of the resonance phenomenon.

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), $\nu = 0.3$.

C. PREPROCESSING OF FEA MODEL

C.1 Creating and saving the project

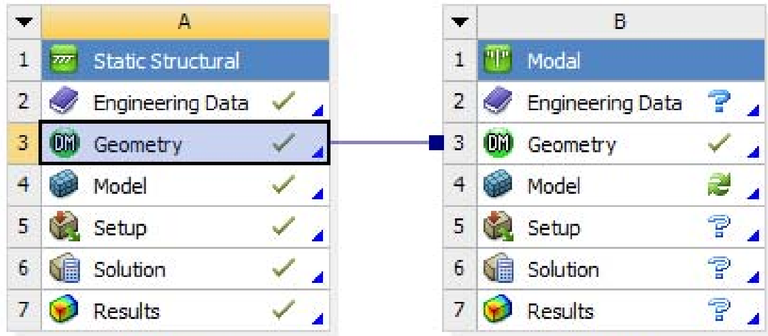
<u>*Creating the project*</u>

The drawing made in the previous application "*Linear static analysis of bar structures*" will be taken over.

In order to take over a geometry previously made in another analysis, the following commands will be executed, in the order presented:

→ Open... (navigate to Windows Explorer and select the previously made "Farm 1D" file) → ↵↵ Modal → will get a window with two analysis structures (Static Structural and Modal).

By holding down Geometry on the command in the Static Structural analysis structure, it is dragged over the command Geometry in the Modal structure until it turns into Share B3, when the mouse key is released OK. The link between the two projects is shown in the figure below → Save As... → File name: *Modal_ex* → Save.



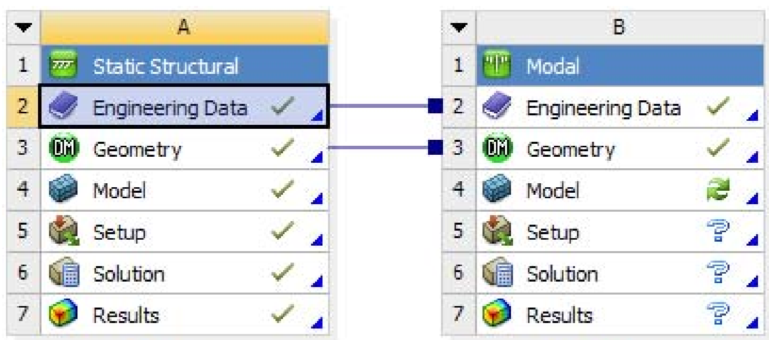
Ferma 1D AEF-A.4.12.

Problem type setting (3D)

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

C.2 Modelling of material characteristics

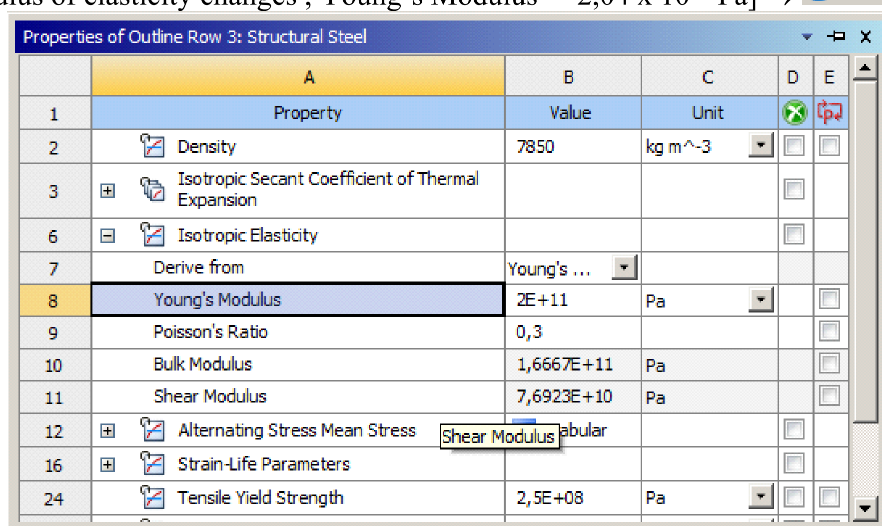
The material of the new structure can be introduced by taking over the characteristics from the main window, with the "pull" procedure from *Static Structural.*



Ferma 1D AEF-A.4.12.

Changing the value of the modulus of elasticity can be done following the commands:

→ ↳ Engineering Data → Edit... → Properties of Outline Row 3: Structural Steel [the value of the longitudinal modulus of elasticity changes , Young's Modulus = 2,04 x 10^{11} Pa] → Return to Project.


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		

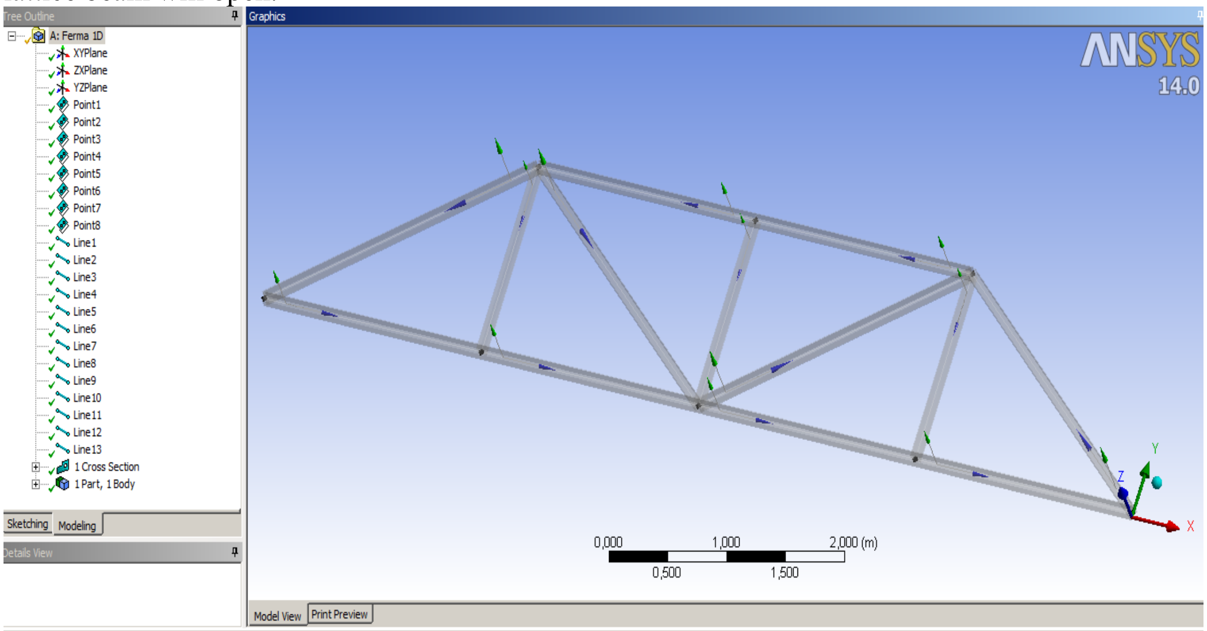
.

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

C.3 Creating the geometric model using DesignModeler (DM)

Making the beam structure with lattice

→ ↲↲ Geometry ✓ from the *Static Structural* analysis structure the geometric model of the lattice beam will open.



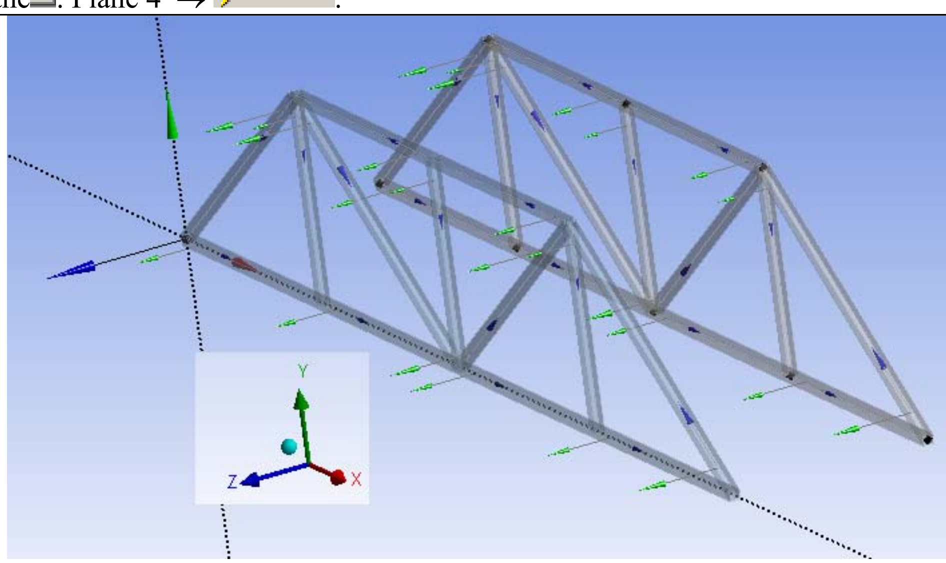
The geometry of the bridge can be continued in two ways: either by building a number of 8 points corresponding to the beam in a plane parallel to the one in which the original model was built (at z = 1.5 m) or by copying the existing beam at distance of 1.5 m on the Oz axis.

Draw a plane parallel to xOy, at a distance of 1.5m.

DM → ↲ New Plane → Details View → **Details of Plane 4** → Type: From Plane → Base Plane: XY Plane → Transform 1 (RMB): Offset Z → FD1, Value 1: 1,5 m → Generate.

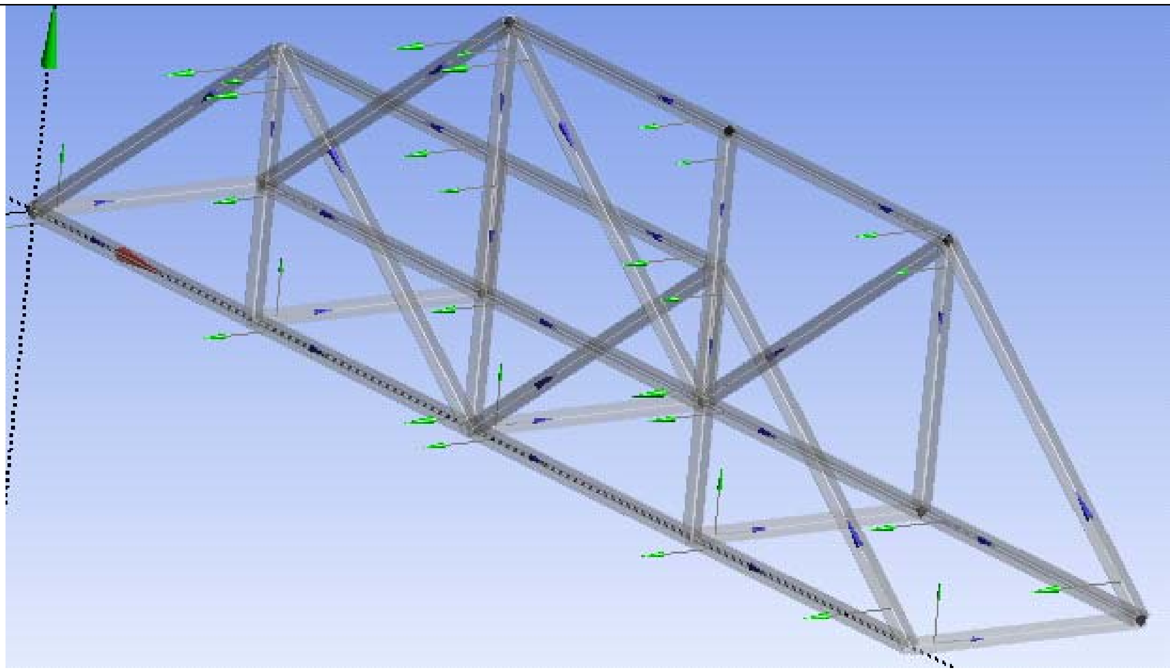
Copy the existing beam into the newly created plane.

DM → Create → Body Operation → Details View → **Details of BodyOp1** → Type: Move → Bodies (select the lattice beam using selection filter) → Preserve Bodies?: Yes → Source Plane: XY Plane → Destination Plane: Plane 4 → Generate.



When there are two parallel lattice beams, the points on the lower base will be joined.

DM → Concept → Lines From Points → the corresponding points are joined → Generate.



Assigning a transverse profile to the metal structure
In terms of section geometry, the newly created segments will have the same properties as the original beam. The beam imported from the previous analysis is assigned a profile, the procedure is as follows:

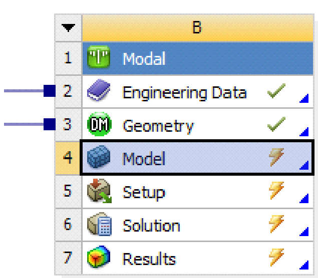
DM → ↲ Line Body → Details View → **Details of Line Body** → Cross Section: Teava_rect_80x80x5; Modeling → View → Cross Section Solids [the 3D section view option is activated] → check that the profiles are oriented correctly; the profile orientation is displayed using green arrows. If the profile is 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.

Saving the geometric model
DM → (Save Project) → File → Close Design Modeler.
→ ↳ Geometry → Properties → Properties of Schematic A3: Geometry → **Advanced Geometry Options** → Analysis Type: 3D → Save.

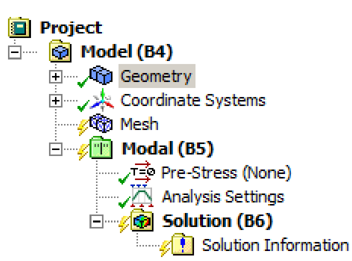
C.4 Finite element modelling

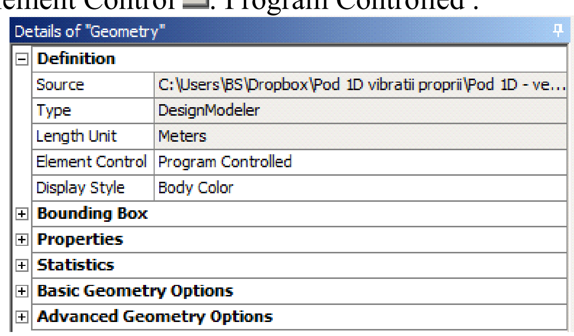
C.4.1 Activate the discretization module and set the finite element type

→ ↲↲ Model → is launched from the analysis structure *Modal, the* modul *Mechanical [ANSYS Multiphysics]*.
↲ Geometry → Details of "Geometry" → **Definition** → Element Control: Program Controlled .



AEF-A.4.12.



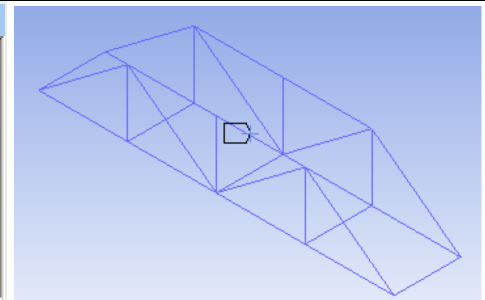


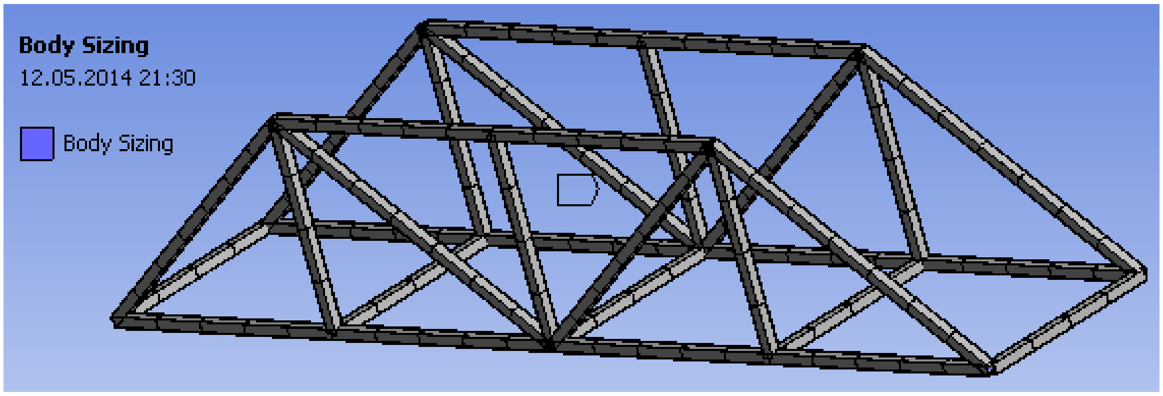
C.4.2 Model discretization and finite element size setting

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

Details of "Body Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	1 Body
Definition	
Suppressed	No
Type	Element Size
Element Size	Default
Behavior	Soft





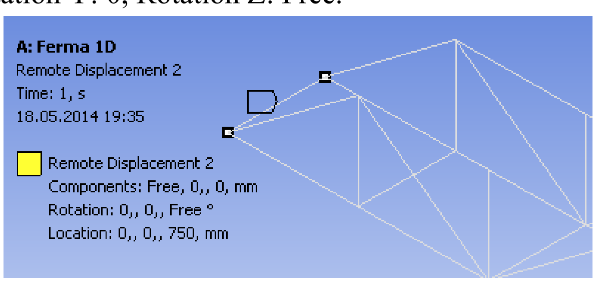
C.5 Supports and restraints modelling

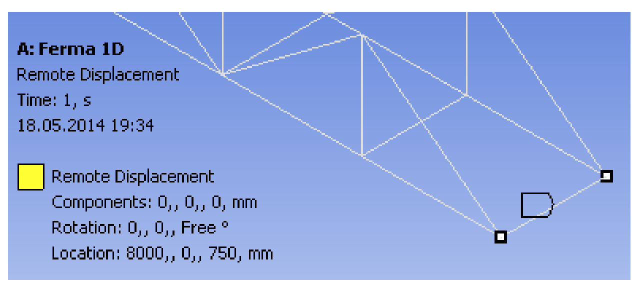
Input restraint

→ Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" → **Scope** → Geometry: [will select, with ↲, holding down the Ctrl key, points P1 and P1', using the selection filter (Vertex)] → Apply; **Definition** → X Component: 0, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.

Will repeat the actions for points P5 şi P5':

→ Static Structural (B5) → Supports → Remote Displacement → Details of "Remote Displacement" → **Scope** → Geometry: [will select, with ↲, holding down the Ctrl key, points P5 and P5', using the selection filter (Vertex)] → Apply; **Definition** → X Component: Free, Y Component: 0, Z Component: 0, Rotation X: 0, Rotation Y: 0, Rotation Z: Free.

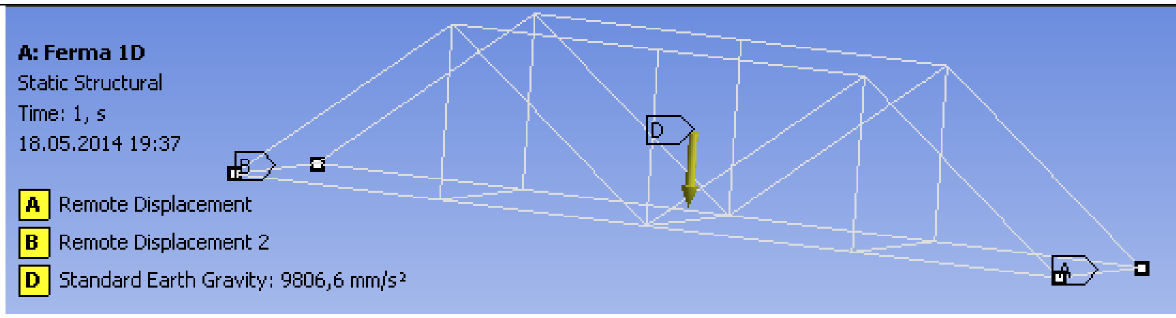




C.6 Load modeling

Introduction of gravitational acceleration

→ 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.



D. SOLVING THE FEA MODEL

D.1 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 command box open with ↳].
The same result can be obtained by using commands:
↲ Solution (B6) → Deformation → Total [the buttons in the menu bars are used]

In order to obtain suggestive results, analyzes will be performed with several types of profiles of the beams of the metal structure: rectangular pipes with dimensions 80 x 80 x 5 mm, 60 x 60 x 5 mm, 50 x 50 x 5 mm and profiles I of sections of 3800 mm^2, 950 mm^2, 237.5 mm^2.

D.2 *Launching the solving module*

→ ↲ Solution (B6) → Solve.

Solution (B6)
Solution Information
Total Deformation
→ Solve →
Solution (B6)
Solution Information
Total Deformation

E. POST-PROCESSING OF RESULTS

E.1 Viewing the displacement fields

For suggestive results, set the view scale of the menu bars:
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.

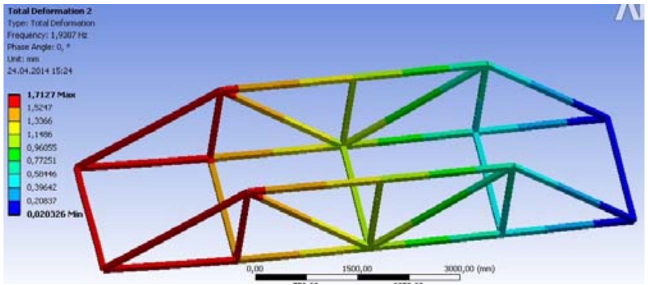
The display characteristics can be changed: the number of frames 10 Frames, as well as the running time of the simulation. At the same time, the result can be saved as a video file using the *Export Video File* command .

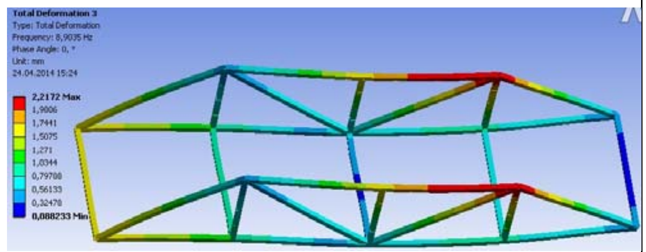
The following are some results of the values of eigenfrequencies and vibration modes for the various profiles analyzed.

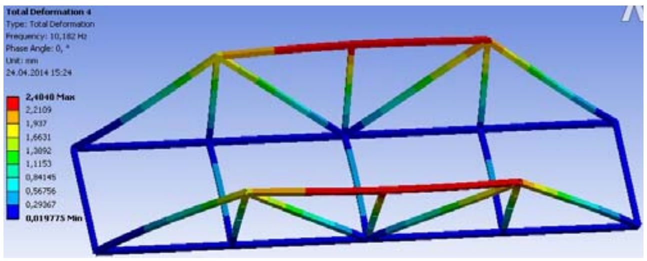
Rectangular pipe 80 x 80 x5 mm

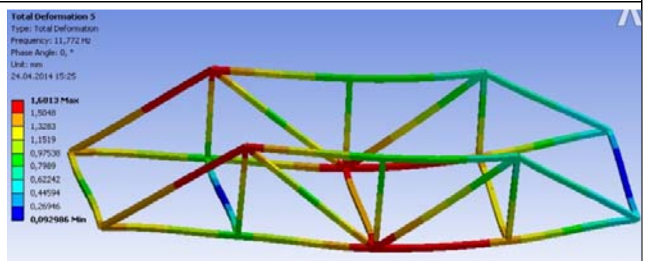
Volume	9,45e+007 mm³
Mass	741,82 kg
Length	63000 mm
Cross Section	Teavarectangulara80x80x5
Cross Section Area	1500, mm²

	Mode	Damped Frequency [Hz]
1	1,	2,1807e-005
2	2,	2,932e-005
3	3,	1,9387
4	4,	8,9035
5	5,	10,182
6	6,	11,772





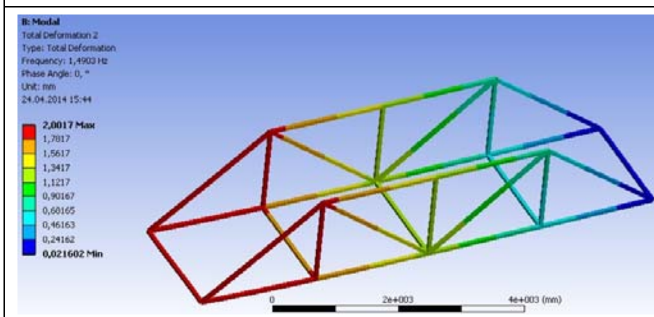


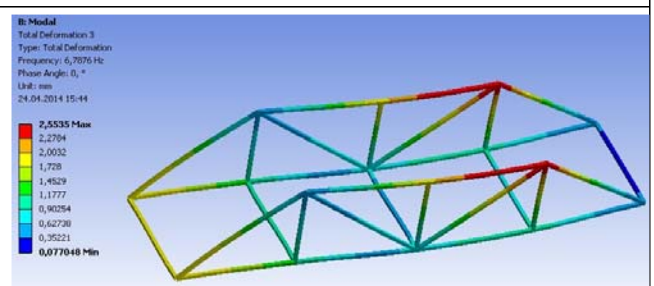


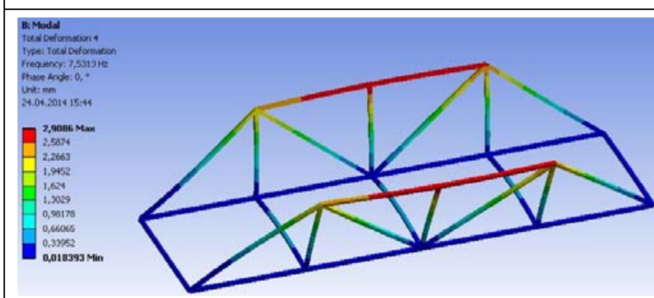
Rectangular pipe 60 x 60 x 5 mm

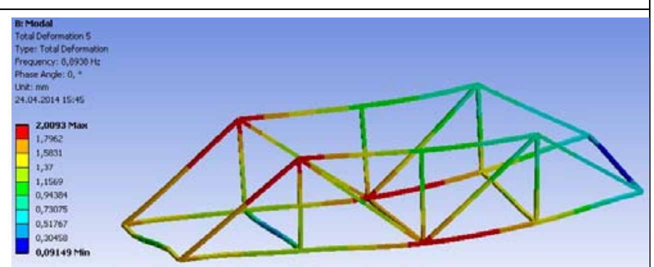
Volume	6,93e+007 mm³
Mass	544,01 kg
Length	63000 mm
Cross Section	Teavarectangulara60x60x5
Cross Section Area	1100, mm²

	Mode	Damped Frequency [Hz]
1	1,	1,9005e-005
2	2,	2,9813e-005
3	3,	1,4903
4	4,	6,7876
5	5,	7,5313
6	6,	8,8938





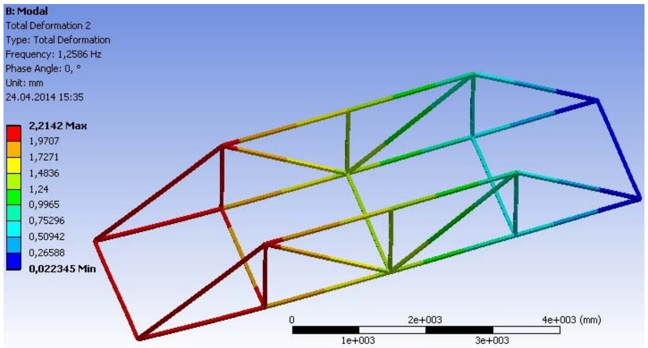


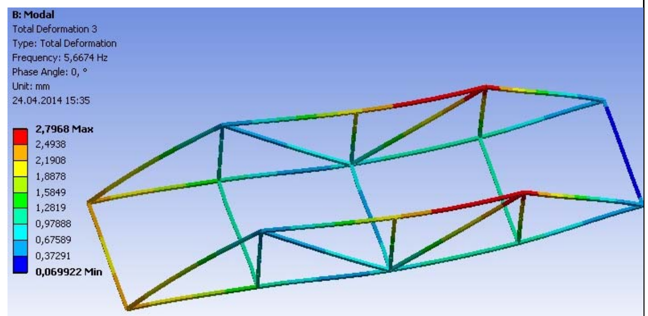


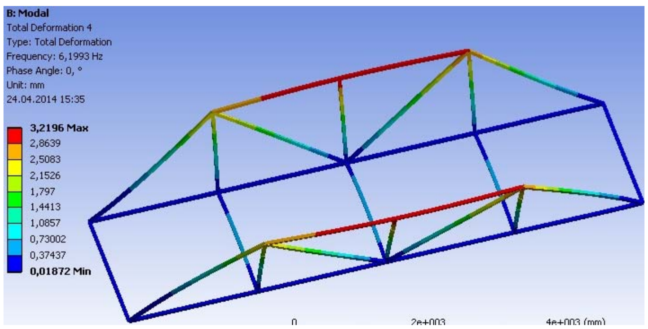
Rectangular pipe 50 x 50 x 5 mm

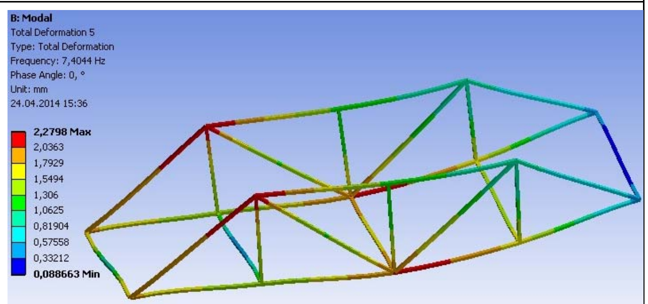
Volume	5,67e+007 mm³
Mass	445,1 kg
Length	63000 mm
Cross Section	Teavarectangulara50x50x5
Cross Section Area	900, mm²

	Mode	Damped Frequency [Hz]
1	1,	7,3932e-006
2	2,	2,6739e-005
3	3,	1,2586
4	4,	5,6674
5	5,	6,1993
6	6,	7,4044





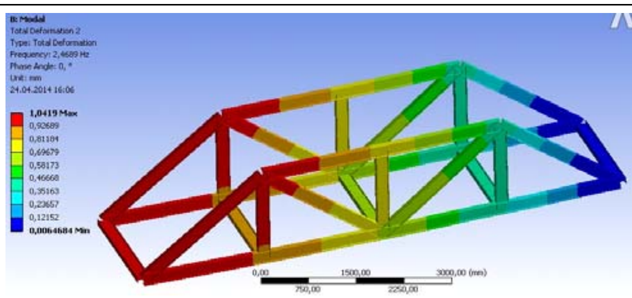


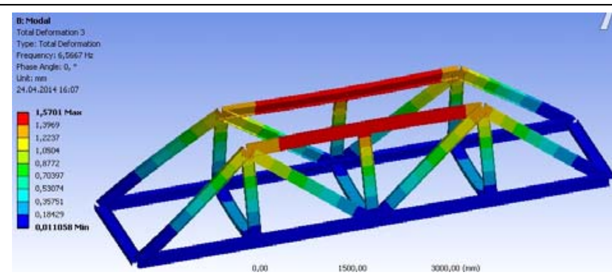


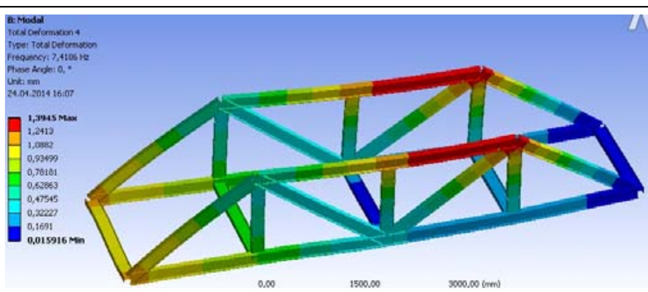
Profil I1, secțiune 3800 mm²

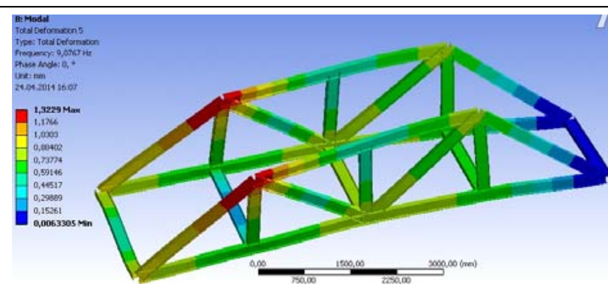
Volume	2,394e+008 mm³
Mass	1879,3 kg
Length	63000 mm
Cross Section	I1
Cross Section Area	3800, mm²

Mode	Damped Frequency [Hz]
1,	0,
2,	0,
3,	2,4689
4,	6,5667
5,	7,4186
6,	9,0767





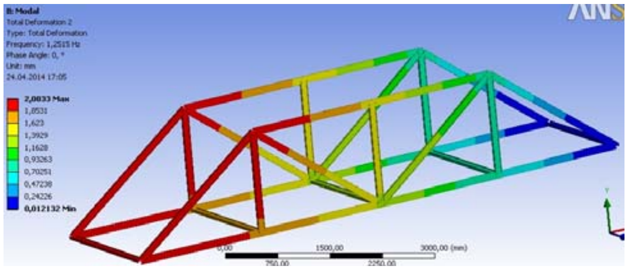


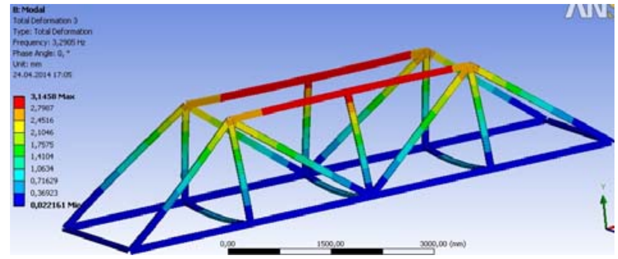


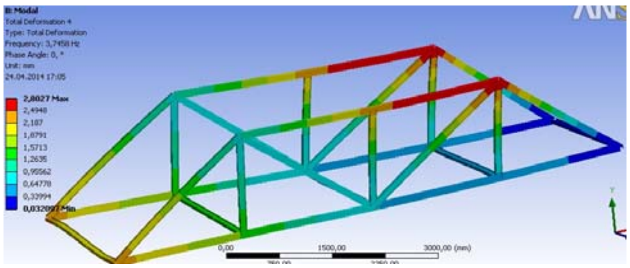
Profil I2, secțiune 950mm²

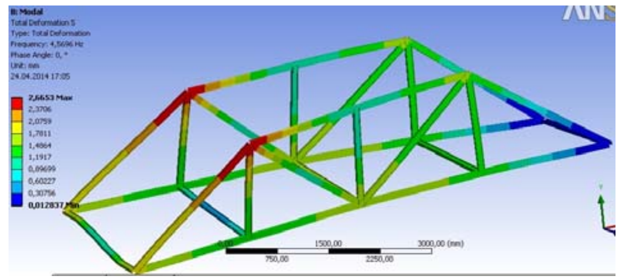
Volume	5,985e+007 mm³
Mass	469,82 kg
Length	63000 mm
Cross Section	I2
Cross Section Area	950, mm²

Mode	Damped Frequency [Hz]
1,	0,
2,	0,
3,	1,2515
4,	3,2905
5,	3,7458
6,	4,5696





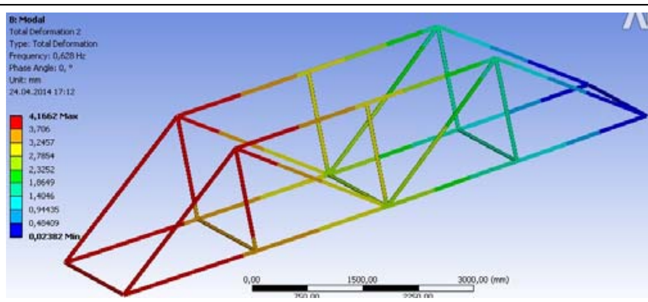


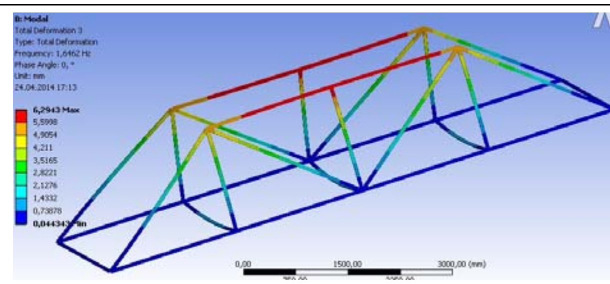


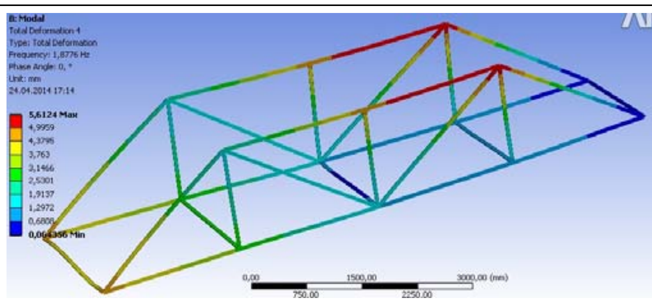
Profil I3, secțiune 237,5 mm²

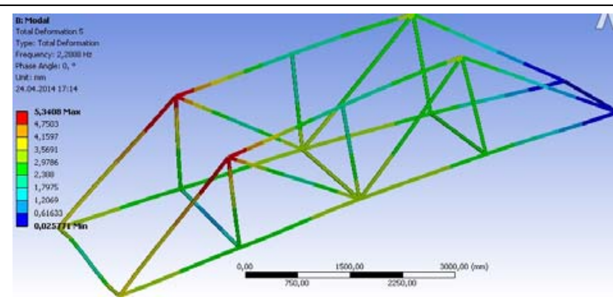
Volume	1,4962e+007 mm³
Mass	117,46 kg
Length	63000 mm
Cross Section	I3
Cross Section Area	237,5 mm²

Mode	Damped Frequency [Hz]
1,	0,
2,	0,
3,	0,628
4,	1,6462
5,	1,8776
6,	2,2888





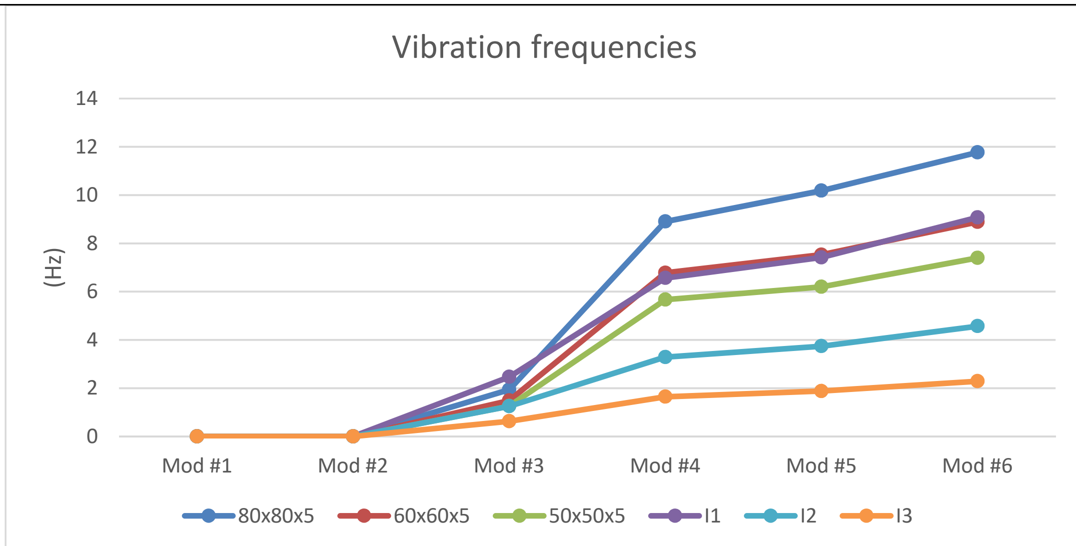




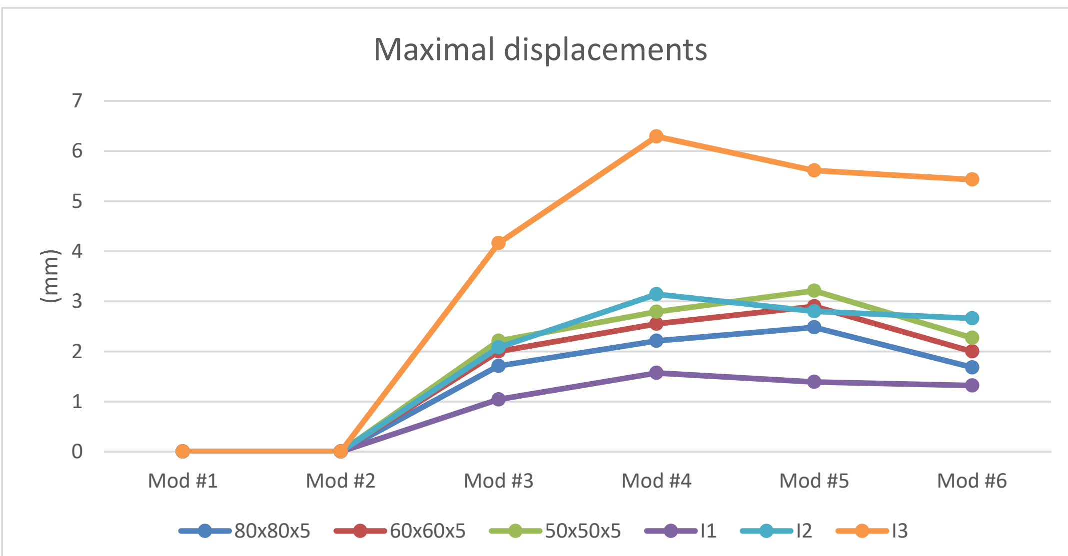
F. ANALYSIS OF RESULTS

It is observed that, despite the fact that the modeling of the bar 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 recorded own frequencies, it can be concluded that with the increase of the beams section, the value of the frequencies will increase, regardless of the transversal profile used. This is observed for both types of profiles analyzed: rectangular profile and for profile I.

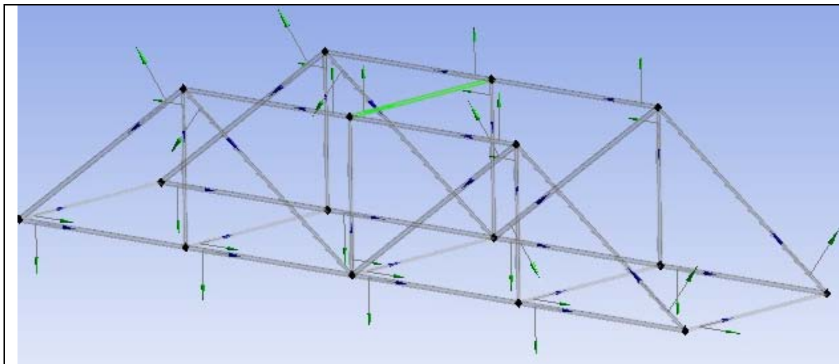


For equivalent profiles in terms of the value of the section surfaces, it can be seen that the rectangular profile generates its own vibrations with higher frequencies than the I profile.

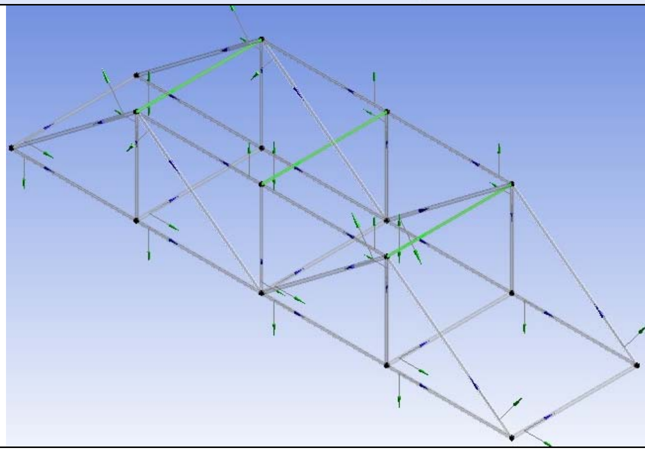


From the point of view of the total displacements, it is observed that the maximum values are found in the own vibration modes Mode # 4 or Mode # 5, after which they decrease with the increase of the own vibration frequencies.

Some applications aim to increase the rigidity of a structure or change its own frequencies, based on an existing structure. In this case, for the structure built from profile I3 (with the smallest cross section of the analyzed ones) two modifications are considered: the installation of some sleepers (case 1 - a sleeper, case 2 - 3 sleepers) in the upper part, as follows . It is observed after the analysis of the vibration modes that the values of the own frequencies have changed compared to the original structure in the direction of the increase.



Mode	Damped Frequency [Hz]
1,	1,027e-005
2,	0,
3,	0,63347
4,	1,9132
5,	2,4412
6,	2,7407



	Mode	Damped Frequency [Hz]
1	1,	0,
2	2,	0,
3	3,	0,66262
4	4,	1,9957
5	5,	2,5659
6	6,	4,0492

G. CONCLUSIONS

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 bar 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 bar structure are controllable processes, which can be done automatically or manually.

The modal analysis of a lattice beam structure is a relatively simple activity, and various modifications of the structure can be made depending on the objectives pursued. Changing the profile of the beam sections and recalculating is done in a very short time.