



LINEAR STATIC ANALYSIS OF TRUSS

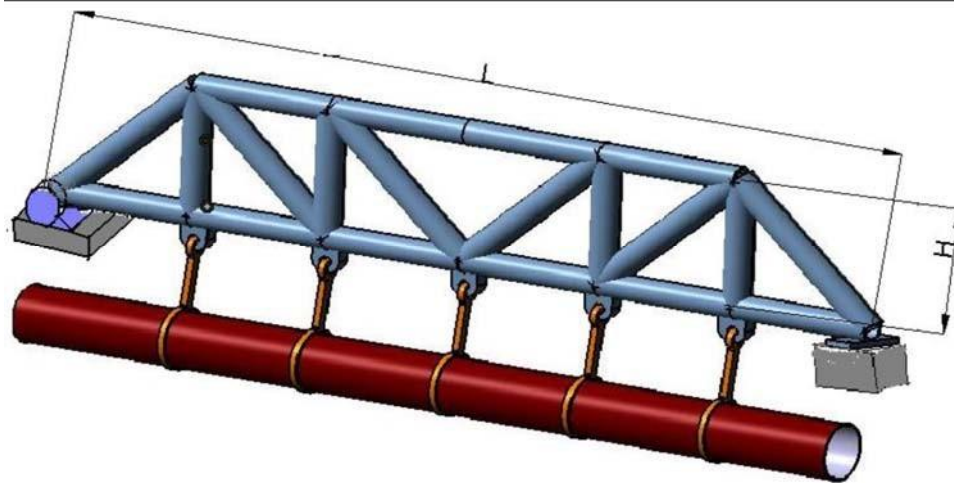
1

INTRODUCTION

The beams, as semi-finished with one of the dimensions much larger than the other two, have different constant sections (circular, square, rectangular, profiles, etc.). Structures made of beams are specific to metal constructions (bridges, pillars, farms, etc.). For finite element analysis, the beam structures are modeled with one-dimensional finite elements whose properties are determined by dimensional and orientation sectional parameters. By these modeling, the memory requirements as well as the calculation time are substantially reduced. The results obtained from the analyzes with these finite elements are less valid in nodal bonding areas (welds, riveting assemblies, screw assemblies) that can be analyzed separately using 3D finite elements and bonding

DESCRIPTION OF THE APPLICATION

In order to support a water supply pipeline when crossing a river, it is necessary to make a lattice beam structure. The pipe is attached to the beams on the beam, placed at equal intervals by means of fixing brackets. In order to avoid thermomechanical stresses at temperature variations, the beam is fixed at one end by means of a bolt assembling which allows rotation and at the other end is supported and guided to allow translation.

**PURPOSE OF THE APPLICATION**

In this case, it is presented the analysis of the fields of displacements, deformations and tensions of a static structure in order to optimize its construction, respectively to minimize its weight under observance of deformation and resistance restrictions. For the beginning of the finite element analysis, the supporting structure in the above figure is considered to be made of OL37 steel pipe with diameters in the interior, $D_i = 70$ mm, and outside, $D_e = 75$ mm. The dimensions of the supporting structure are: $L = 60$ m, $H = 10$ m. For finite element analysis, the action of the supported pipe on the structure can be modeled by inserting into each node where the pipe is clamped by means of the clamps of a concentrated force $F = 100$ kN. In addition, it is of particular importance for the analysis of these structures, also, to take into account the internal weight of their own weight.

FEA Application A.2.1

LINEAR STATIC ANALYSIS OF TRUSS BEAM STRUCTURES

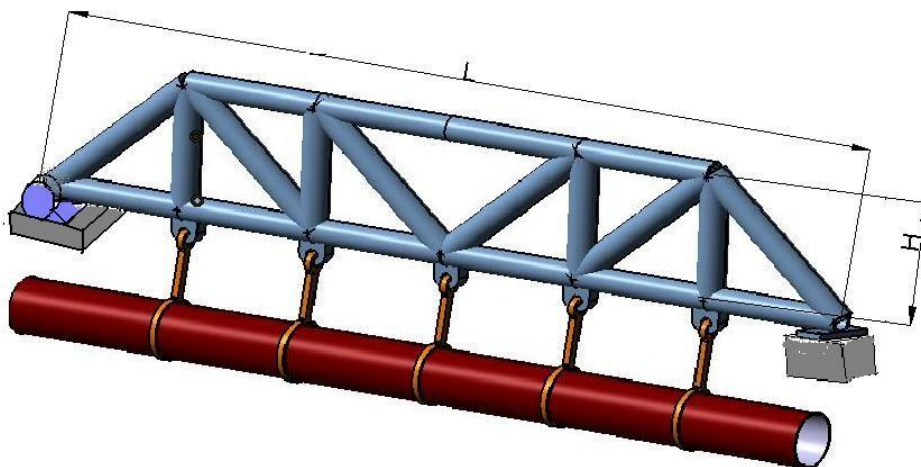
A. DEFINING THE APPLICATION

A.1. Introduction

Bars, as primary semi-finished products with one dimension much larger than the other two, have various constant cross-sections (circular, annular, square, rectangular, profiles, etc.). Structures made from bars are specific, primarily, to metal constructions (bridges, beams, columns, trusses, etc.). For finite element analysis, bar structures are modeled with one-dimensional finite elements whose properties are determined by dimensional and orientation sectional parameters. These models substantially reduce memory requirements as well as calculation time. The results obtained from analyses with these finite elements are less valid in the nodal connection zones (welds, riveted joints, bolted joints), which can be analyzed separately using 3D and connection finite elements.

A.2. Description of the Application

To support a water supply pipeline crossing a river, a truss-type beam structure must be built. The pipeline is attached to supports on the beam, placed at equal intervals, using fixing clamps. To avoid thermomechanical stresses during temperature variations, the beam is fixed at one end by means of a pin assembly that allows rotation, and at the other end, it is supported and guided, allowing translation.



A.3. Purpose of the Application

This application presents the analysis of the displacement, strain, and stress fields of a statically loaded bar structure in order to optimize its construction, specifically to minimize its weight while respecting deformation and strength constraints. For the beginning of the finite element analysis, the support structure in the figure above is considered to be made of OL37 steel pipe with an inner diameter, $D_i = 70mm$, and an outer diameter, $D_o = 75mm$. The dimensions of the support structure are: $L = 60m$, $H = 10m$. For the finite element analysis, the action of the supported pipe on the structure can be modeled by introducing a concentrated force $F = 100KN$ at each node where the pipe is attached via fixing clamps. In addition, of particular importance for the analysis of these structures is taking into account internal forces such as self-weight.

B. PREPARING THE ANALYSIS MODEL

B.1. Defining the Analysis Model

To prepare the finite element analysis model associated with the application above, it is necessary to identify:

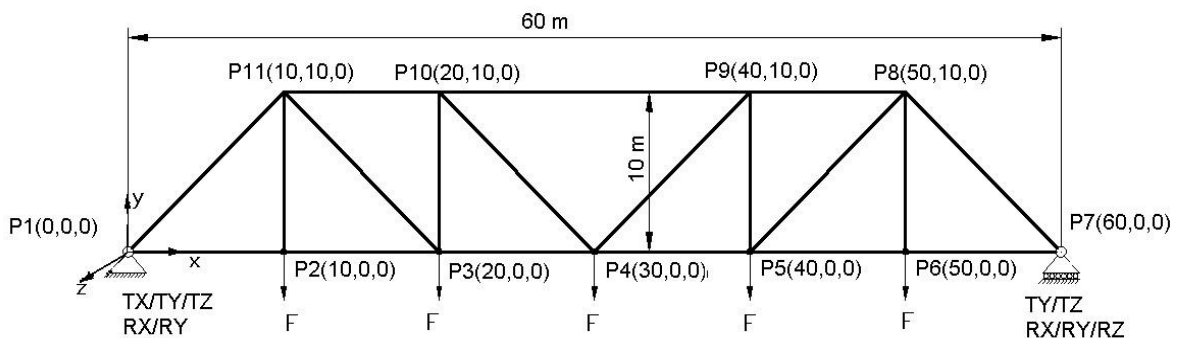
- The geometric shape and dimensions,
- Constraints induced by connections with adjacent elements,
- External and internal (self-weight) loads,
- Material characteristics.

B.2. Description of the Analysis Model

The geometric shape and dimensions of the support structure's analysis model are identical to those of the structure at the level of the segment axes. For the analysis, the structure is modeled with 1D finite elements, and thus, the geometric model has the configuration shown in the figure below.

For the analysis model to behave the same as the real model, it is necessary to associate boundary conditions that involve canceling translational displacements and rotations relative to the OX and OY axes at point P1, and rotations and translations along the OY and OZ axes at point P7.

The structure of the analysis model is loaded with a concentrated force $F = 100KN$ at points P2 – P6.



B.3. Establishing Material Characteristics

For the finite element analysis, the strength characteristics of the material, OL 37 steel, are:

- Longitudinal modulus of elasticity, $E = 20400N/mm^2$
- Transverse contraction coefficient (Poisson's ratio), $\nu = 0.3$

C. PREPROCESSING THE ANALYSIS MODEL

C.1. Geometric Modeling

Activate the solid generation module and set the unit of measurement for lengths

Start → Mechanical Design → Part Design → New part: New part name: **Grinda**.

Tools → Options... → Options: Parameters and Measure; Units; Length, **Millimeter (mm)**; **OK**.

Generate the reference sketch

 (Point) → Point Definition: X 0mm, Y 0mm, Z 0mm; **OK**. [similarly, the coordinates of the other points are entered (fig. B2): P2(40,0,0), P3(20,0,0), P4(20,-7.5,0), P5(35,7.5,0), P6(0,-30,0), P7(-35,-7.5,0), P8(-20,-7.5,0), P9(-20,0,0), P10(-40,0,0), P11(-50,0,0), P12(-50,-100,0)].

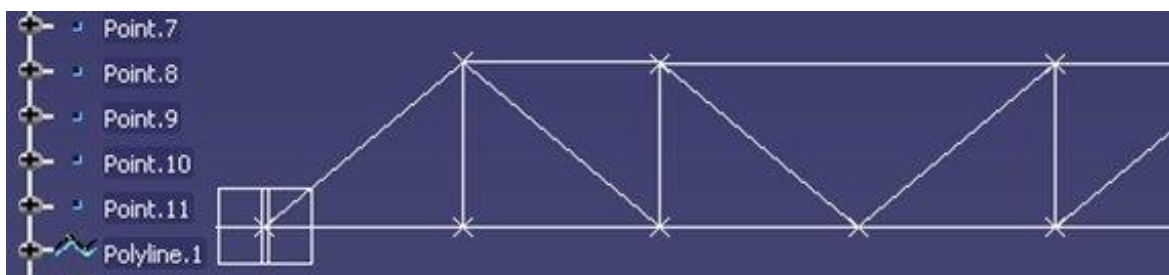


Generate the bar structure

 (Polyline) → Polyline Definition [select with the mouse from the graphical area or from the tree structure the points: Point.2, Point.3 ... Point.11, Point.1, (simultaneously generating their list in the Polyline Definition window)]; **Close Polyline**, [select with the mouse Point.1 from the Polyline Definition list], **Add after**, [select with the mouse from the graphical area or from the tree structure the points, in the following order: Point.2, Point.11, Point.3, Point.10, Point.4, Point.9, Point.5, Point.8, Point.6]; **OK** → Mult Result Management: **Keep all the sub-elements**, **OK**.

Observation

If the (Polyline) button is not found in the command area (Toolbar), the following command sequence will be executed: View → Toolbars → Customize... Customize: Commands, Categories ↓ All Commands, Commands ↓ Polyline... → place the command symbol with the mouse (Drag Drop) in the command bar area (Toolbar), **Close**.



C.2. Modeling the Material Characteristics

[Select the PartBody specification] → (Apply Material) → Library (ReadOnly): Metal, Steel, Apply Material; **OK**.

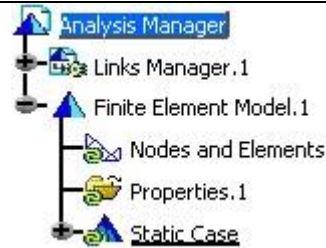
[Select the Steel specification from the tree structure] → Properties: Feature Properties, Feature Name: OL37; Analysis, Young Modulus 204000 N/mm², Poisson Ratio 0.3; Apply; **OK**.



C.3. Finite Element Modeling

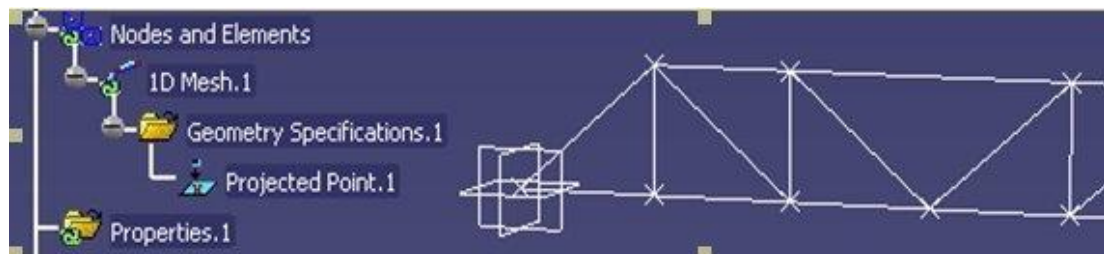
Activate the advanced finite element meshing module

Start → Analysis & Simulation → Advanced Meshing Tools → New Analysis Case: Static Analysis, ↵OK.



Enter the finite element type and discretization characteristics

(Beam Mesher) → Beam Meshing: [select the polyline, Polyline.1], Element size: 200 mm, (Add isolated points) → External Point Constraint: Supports: [select with the mouse from the graphical area or from the tree structure the points: Point. 1,... Point.7]; Tolerance 0 mm, ↵OK → ↵OK.

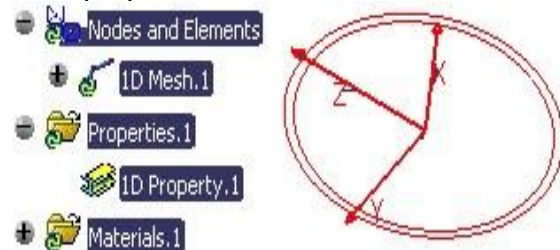


Activate the finite element analysis module

Start → Analysis & Simulation → Generative Structural Analysis.

Model the finite element properties

(1D Property) → 1D Property: Supports: [select from the tree structure, 1DMesh.1]; Type ↓ Tubular beam, → Beam Definition: Outside radius: 75 mm, Inside radius: 70 mm, ↵OK → ↵OK.

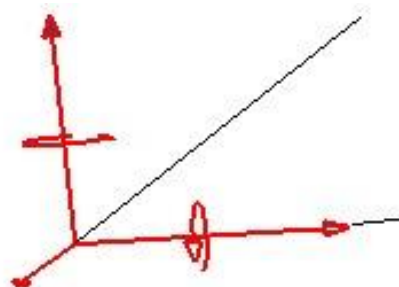


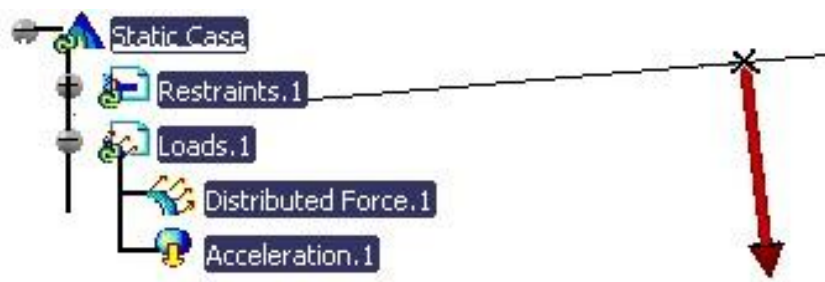
C.4. Modeling Geometric Constraints

Introducing displacement restrictions

(User-defined Restraint) → User-defined Restraint: Supports: [select point P1, Point.1 with the mouse], ☐ Restrain Rotation 3 →OK.

(User-defined Restraint) → User-defined Restraint: Supports: [select point P7, Point.7 with the mouse], ↵OK. ☐ Restrain Translation 1 , →OK.

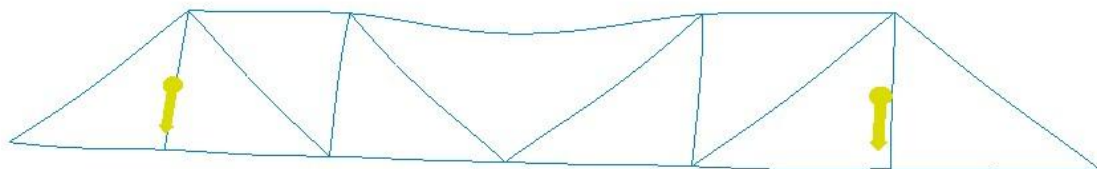
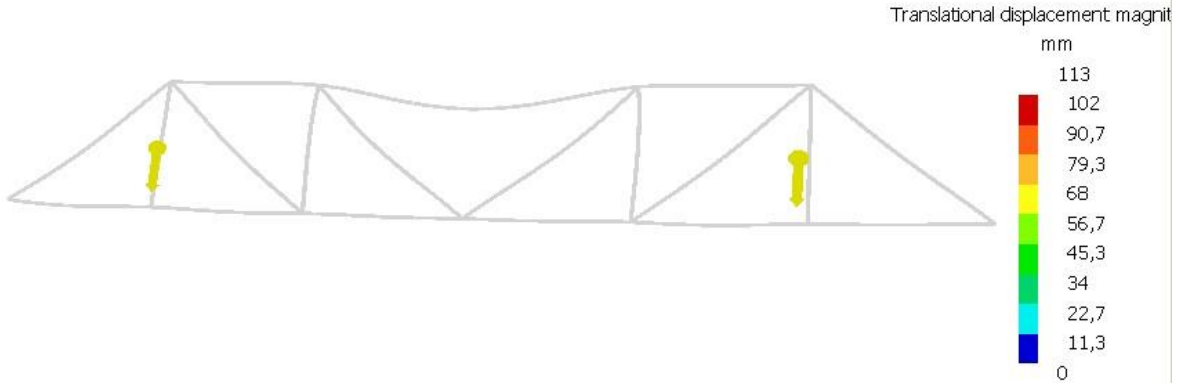


C.5. Modeling Loads	
<i>Introducing the forces</i>	
	(Distributed Force) → Distributed Force: Supports [select points P2...P6 with the mouse]; Force vector X 0N, Y – 100 kN, Z 0N; ↵OK.
	(Acceleration) → Acceleration: Supports [select the 1D Mesh.1 specification with the mouse]; Acceleration Vector X 0 m/s ² , Y -9.8 m/s ² , Z 0 m/s ² ; ↵OK.
	

D. SOLVING THE FINITE ELEMENT MODEL

D.1. Launching the Calculation Module	
	(Compute) → Compute: ↓ All, ↵OK → Computation Resources Estimation, ↵Yes → Computation Status.

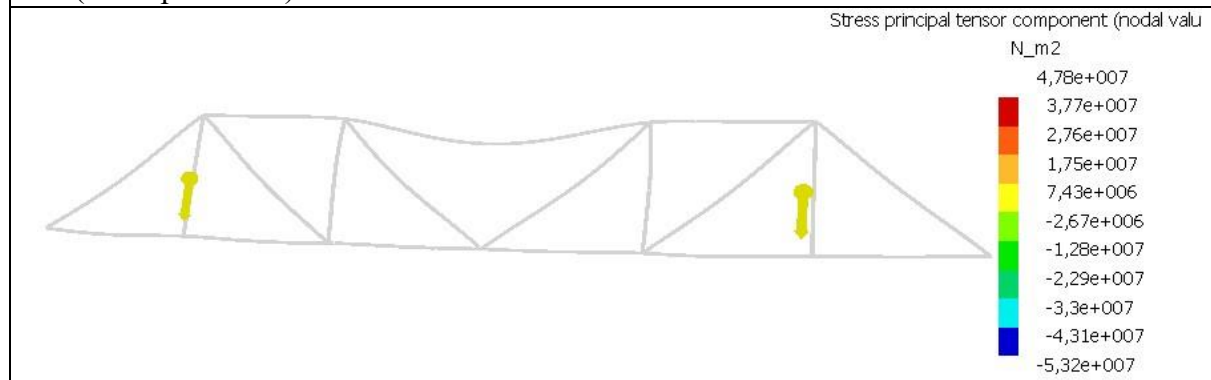
E. POSTPROCESSING THE RESULTS

E.1. Visualizing the Deformed State/Animation	
	(Deformation).
	(Animate) → Animate: Steps number: 20 (other parameters can also be modified), ↵Close.
	
E.2. Visualizing the Displacement Field	
	(Displacement).
	

E.3. Visualizing the Stress Field



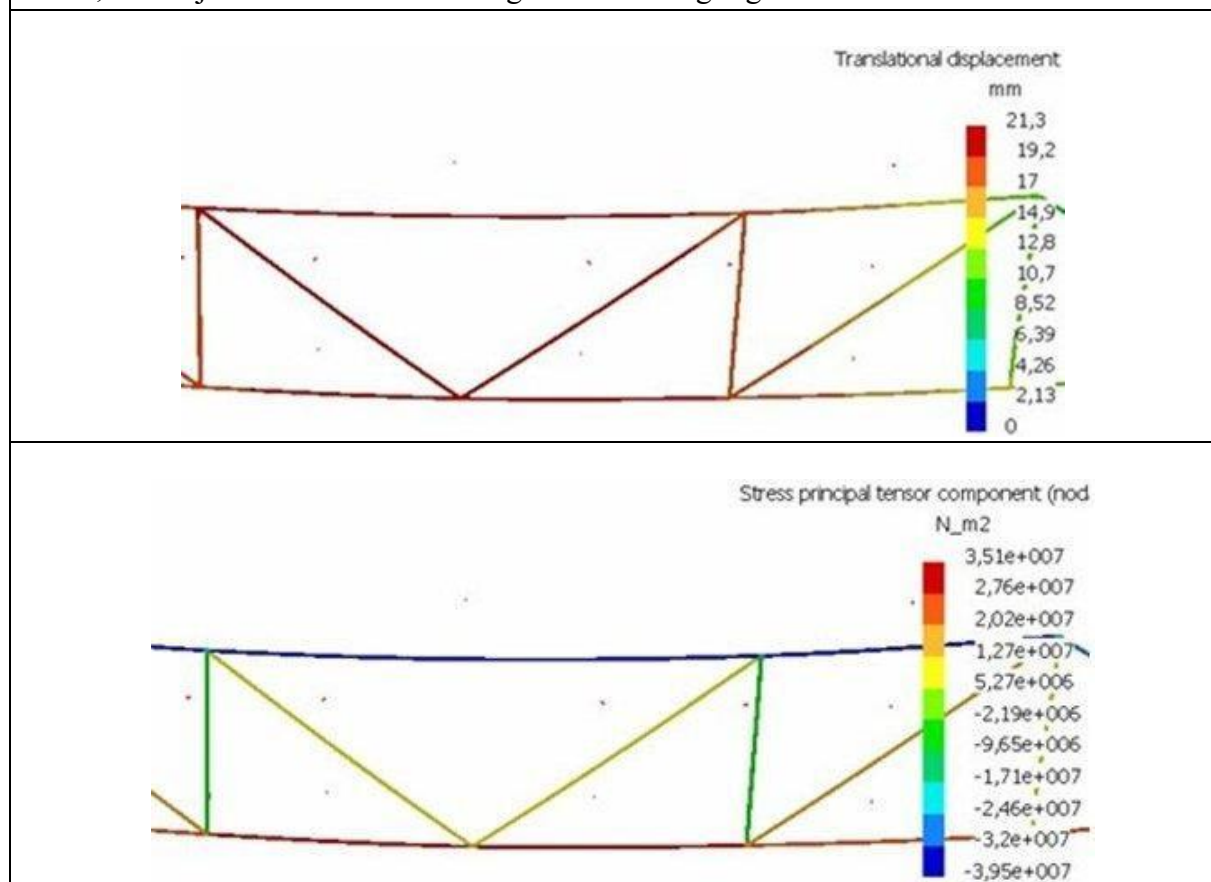
(Principal Stress).



F. Conclusion

From the analysis of the displacement and stress fields, it is evident that the analyzed element is a quasi-rigid structure, with reduced maximum displacements in the vertical direction, OY, 113 mm in the middle area of the structure at point P4, where the maximum compressive stress also has a value of 53.2 MPa. The maximum tensile stress is 47.8 MPa at point P4. These stresses are lower than the admissible stresses, 90 - 120 MPa (yield strength approx. 330 MPa).

The figures below show the displacement and stress fields for the case where self-weight forces are not considered, by re-analyzing the model without the Acceleration.1 load. Thus, observing that the maximum displacement is 5 times smaller and the stresses are over 20% lower, the major effect of the self-weight forces is highlighted.



G. EXERCISE

Perform the analysis of the structure in the adjacent figure, representing a metal gate with dimensions:

$L = 3\text{ m}$, $H = 1.5\text{ m}$, made of rectangular OL37 steel tubing with dimensions $a = 50$, $b = 30$ and thickness 3 mm . The analysis will also take into account the self-weight of the materials.

