FACULTE ENVIRONNEMENT NATUREL, ARCHITECTURAL ET CONSTRUIT
ENAC
INSTITUT D'INGÉNIERIE CIVILE IIC
Laboratoire des Structures Métalliques Résilientes
RESSLAB

Téléphone : +41 21 693 24 27
Fax : +41 21 693 28 68
E-mail : dimitrios.lignos@epfl.ch
Site web : http://resslab.epfl.ch
Address: EPFL ENAC IIC RESSLAB
GC B3 485, Station 18,
CH-1015, Lausanne

In-class Exercise Week #2: Matrix structural analysis in a programming environment

Exercise #1:

For the first exercise of the first week, write a basic program in any programming language that computes the magnitude and direction of the force P at point a required to displace that point a vertically downward 5mm without any horizontal displacement. Compute the member forces and the deflected shape of the truss frame shown in the figure.

- Assume that $E = 200\ GPa$.
- Assume a linear static analysis.

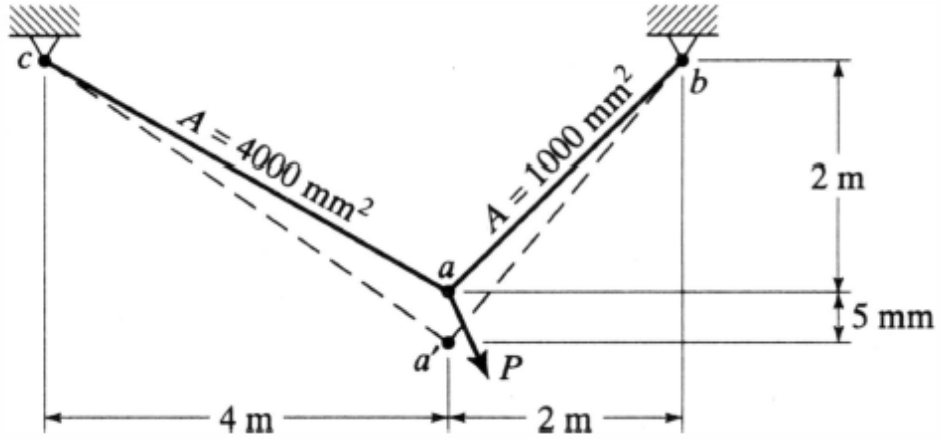

Figure 1. Schematic representation of the truss frame when subjected to the force P

Basic workflow for the solution (to be posted after releasing the solution of assignment #1):

The following steps can be used to write a program to solve this problem:

1. Define the structure:
   1.1. Coordinates of the nodes in a global coordinate system
   1.2. Connectivity of the structure
   1.3. Geometric and material properties of each element
   1.4. Define a mapping between the local degrees of freedom at each node and the global degrees of freedom
   1.5.
2. Assemble the global stiffness matrix. For each element:
   2.1. Compute the element stiffness matrix in the local coordinate system
   2.2. Compute the element stiffness matrix in the global coordinate system using rotation matrices
   2.3. Using the defined mapping and the element stiffness matrix in the global coordinate system, assemble the global stiffness matrix

3. Determine the fixed and the free global degrees of freedom

4. Assemble the global force vector and the global displacement vector:
   4.1. Initialize the vectors as vectors of 0
   4.2. Add the imposed boundary conditions (imposed nodal displacements and/or nodal forces)

5. Solve the global force-displacement equation:
   5.1. To solve for the nodal displacement knowing the nodal forces, perform the static condensation

6. Determine the member forces (this step depends on the element type). For truss elements:
   6.1. Compute the nodal displacements in the local reference frame
   6.2. Compute the member force using the element stiffness matrix in the local reference frame

For Exercise 1, the following element forces are obtained:

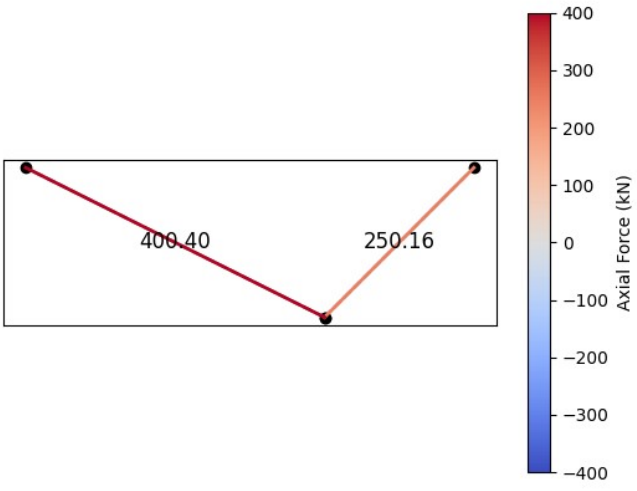

Figure 1 - Member forces