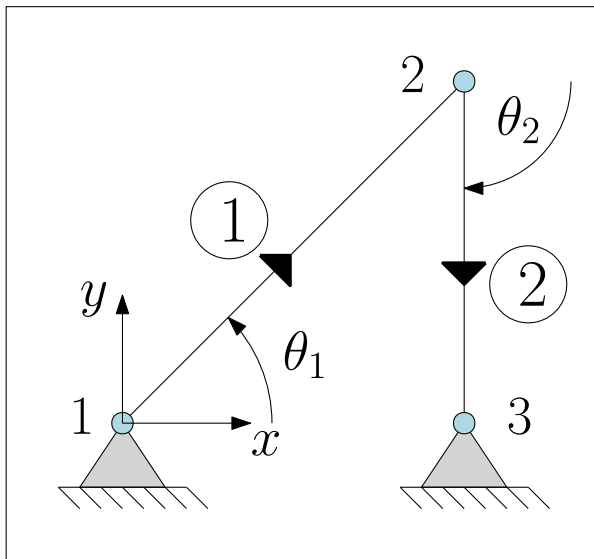


Mesh structure

The structure is composed of two bars. There are two degrees of freedom for each node : the displacement along the horizontal and vertical direction. The structure is isostatic and the bars only work in traction-compression.

The node are numbered according to the following picture:



The global reference frame is located in correspondance of node 1.

Two information are required to construct the mesh

1. The table of nodal coordinates is the following

Global node number	x	y
1	0	0
2	L	L
3	L	0

2. The table of connectivity

Element	Left node (1)	Right Node (2)
1	1	2
2	2	3

The table of connectivity describes the topology of the mesh, i.e. how the elements are associated to the nodes and how they are oriented.

The orientation provides the information on the angle by which a local frame is rotated with respect to the global frame. The x axis of local frame is aligned along an element and oriented from the left to right node. For this case the angles are $\theta_1 = \pi/4$ and $\theta_2 = -\pi/2$.

N.B. Different types of elements may be contained in a single mesh (1D, 2D, 3D).

The coordinates and connectivity table can be declared by the following code (remember that in python the indexing starts at 0):

```
In [1]: import numpy as np

L = 1 # length given by the data

coordinates = np.array([[0, 0],
                        [L, L],
                        [L, 0]])

connectivity_table = np.array([[0, 1],
                              [1, 2]])
```

Forces and nodal displacement

The vector of nodal displacement will contain the horizontal u_i , $i \in \{1, 2, 3\}$ and vertical v_i , $i \in \{1, 2, 3\}$ displacements. The displacements at nodes 1 and 3 are zero because of the boundary conditions. An external forces is applied on node 2 and reaction forces enforce the boundary conditions on node 1 and 3.

$$\mathbf{q} = \begin{pmatrix} u_1 = 0 \\ v_1 = 0 \\ u_2 \\ v_2 \\ u_3 = 0 \\ v_3 = 0 \end{pmatrix}, \quad \mathbf{f} = \begin{pmatrix} X_1 \\ Y_1 \\ F \\ 0 \\ X_3 \\ Y_3 \end{pmatrix}$$

The finite element system is constructed considering the minimization of the total elastic energy (principle of virtual work):

$$\delta E = \delta W_{\text{ext}}.$$

Upon introduction of the finite element discretization, this translate into the following algebraic system

$$\delta \mathbf{q}^\top \mathbf{K} \mathbf{q} = \delta \mathbf{q}^\top \mathbf{f}, \quad \forall \delta \mathbf{q} \quad \implies \mathbf{K} \mathbf{q} = \mathbf{f}.$$

Local stiffness matrix construction

In this example the energy is the traction deformation energy, given by

$$E_{ax} = \sum_1^{n_{el}} \int_0^{L_e} EA_e \left(\frac{du^l}{dx^l} \right)^2 dx^l$$

where u^l is the axial displacement and x^l is the local coordinate (aligned with the element axis). The axial displacement field is approximated using Lagrange polynomials in the local reference frame as follows

$$u^l(x^l) = \begin{pmatrix} 1 - x^l/L_e & x^l/L_e \end{pmatrix} \begin{pmatrix} u_1^l \\ u_2^l \end{pmatrix} = \mathbf{N} \mathbf{u}^l.$$

where L_e is the length of the element under consideration and l stands for local frame. The local stiffness matrix is computed as

$$\mathbf{K}_e^l = \int_0^{L_e} EA_e \frac{d\mathbf{N}^\top}{dx^l} \frac{d\mathbf{N}}{dx^l} dx^l$$

In finite elements technology, the description of a finite element is typically given in a simple reference configuration. For truss elements, reference element is a segment of unitary length. The reference element coordinate is simply given by $\xi = L_e^{-1} x^l$. The bases functions rewrite as follows

$$u(\xi) = \begin{pmatrix} 1 - \xi & \xi \end{pmatrix} \begin{pmatrix} u_1^l \\ u_2^l \end{pmatrix}$$

The stiffness matrix for a generic element in its local reference frame, is computed using the reference coordinate by taking into account the Jacobian of the transformation ($J = \frac{dx^l}{d\xi} = L_e$) and the chain rule

$$\mathbf{K}_e^l = \int_0^1 EA_e \frac{d\mathbf{N}^\top}{d\xi} \frac{d\mathbf{N}}{d\xi} J^{-1} d\xi = \frac{EA_e}{L_e} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}. \quad \text{This is valid in 1D.}$$

The inverse of the Jacobian appear because of the chain rule in the computation of the derivative of the basis function $\frac{d\mathbf{N}}{dx^l} = \frac{d\mathbf{N}}{d\xi} \frac{d\xi}{dx^l} = \frac{d\mathbf{N}}{d\xi} J^{-1}$.

Since the problem is 2D, both longitudinal and trasversal displacement need to be accounted for. Since this is a truss structure (bars only work in traction compression) there is no need to account for the bending stiffness, (the pivots will not transmit torques).

$$\mathbf{K}_e^l = \int_0^1 EA_e \frac{d\mathbf{N}^\top}{d\xi} \frac{d\mathbf{N}}{d\xi} J^{-1} d\xi = \frac{EA_e}{L_e} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}.$$

From local to global stiffness matrix

To obtain the global stiffness matrix, the local stiffness matrix must be transformed to the global reference frame. This is achieved by considering a rotation from the local coordinates to the global coordinates. If the local frame of element e is rotated by an angle θ_e then the rotation matrix is given by

$$\mathbf{R}_e^{l \rightarrow g} = \begin{bmatrix} \cos(\theta_e) & -\sin(\theta_e) \\ \sin(\theta_e) & \cos(\theta_e) \end{bmatrix}$$

This matrix converts the local to the global coordinates

$$\begin{pmatrix} u^g \\ v^g \end{pmatrix} = \mathbf{R}_e^{l \rightarrow g} \begin{pmatrix} u^l \\ v^l \end{pmatrix}.$$

The left and the right nodes are collected in vector

$$\mathbf{q}_e^l = (u_1^l \quad v_1^l \quad u_2^l \quad v_2^l)^\top.$$

The rotation needs to be applied to both nodes, so two rotation matrices are piled to obtain the displacement of the nodes in the global reference frame

$$\mathbf{q}_e^g = \mathbf{T}_e^{l \rightarrow g} \mathbf{q}_e^l, \quad \text{where} \quad \mathbf{T}_e^{l \rightarrow g} := \begin{bmatrix} \mathbf{R}_e^{l \rightarrow g} & 0 \\ 0 & \mathbf{R}_e^{l \rightarrow g} \end{bmatrix}$$

The conversion from the local to the global stiffness matrix is obtained considering the fact the variation of the energy is the same in both references

$$[\delta \mathbf{q}_e^l]^\top \mathbf{K}_e^l \mathbf{q}_e^l = [\delta \mathbf{q}_e^g]^\top \mathbf{K}_e^g \mathbf{q}_e^g$$

This implies the following transformation rule for the stiffness matrix

$$\mathbf{K}_e^g = \mathbf{T}_e^{l \rightarrow g} \mathbf{K}_e^l [\mathbf{T}_e^{l \rightarrow g}]^\top$$

Developing the computation one obtains

$$\mathbf{K}_e^g = \frac{EA_e}{L_e} \begin{bmatrix} \cos^2(\theta_e) & \cos(\theta_e) \sin(\theta_e) & -\cos^2(\theta_e) & -\cos(\theta_e) \sin(\theta_e) \\ & \sin^2(\theta_e) & -\cos(\theta_e) \sin(\theta_e) & -\sin^2(\theta_e) \\ & & \cos^2(\theta_e) & \cos(\theta_e) \sin(\theta_e) \\ \text{sym.} & & & \sin^2(\theta_e) \end{bmatrix}$$

The following python code compute the stiffness matrix given the coordinates of the two nodes of an element and its axial stiffness.

```
In [70]: def truss_2d_element(coord1, coord2, EA):
        """
        Compute the element stiffness matrix for a 2D truss bar in global coordi
        Function translated to python from the MATLAB code in https://people.duk

        Parameters
        -----
        coord1 : 2*1 array
            x1, y1 coordinates of the first node
        coord2 : 2*1 array
            x2, y2 coordinates of the second node
        EA : float
            axial stiffness of the bar

        Returns
        -----
        K : numpy.ndarray

        """

        L = np.linalg.norm(coord2 - coord1) # length of the bar

        x1, y1 = coord1
        x2, y2 = coord2

        c = ( x2 - x1 ) / L # cosine of bar angle
        s = ( y2 - y1 ) / L # sine of bar angle

        theta = np.arctan2(s, c)

        K = EA/L * np.array([[c**2, c*s, -c**2, -c*s],
                             [c * s, s**2, -c*s, -s**2],
                             [-c**2, -c*s, c**2, c*s],
                             [ -c*s, -s**2, c*s, s**2 ] ] )

        return K, theta
```

Once the global stiffness matrix for each element is obtained the assembly of the global stiffness matrix is obtained by considering the connectivity table.

For instance for element 1, the local nodes correspond to the global nodes 1, 2.

$$\mathbf{K}_1 = \begin{bmatrix} [\mathbf{K}_1^g]_{11} & [\mathbf{K}_1^g]_{12} & [\mathbf{K}_1^g]_{13} & [\mathbf{K}_1^g]_{14} & 0 & 0 \\ [\mathbf{K}_1^g]_{21} & [\mathbf{K}_1^g]_{22} & [\mathbf{K}_1^g]_{23} & [\mathbf{K}_1^g]_{24} & 0 & 0 \\ [\mathbf{K}_1^g]_{31} & [\mathbf{K}_1^g]_{32} & [\mathbf{K}_1^g]_{33} & [\mathbf{K}_1^g]_{34} & 0 & 0 \\ [\mathbf{K}_1^g]_{41} & [\mathbf{K}_1^g]_{42} & [\mathbf{K}_1^g]_{43} & [\mathbf{K}_1^g]_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

where $[\mathbf{K}_e^g]_{ij}$ denotes the ij component of the stiffness matrix of element e .

For element 2 the local nodes correspond to the global nodes 2, 3

$$\mathbf{K}_2 = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & [\mathbf{K}_2^g]_{11} & [\mathbf{K}_2^g]_{12} & [\mathbf{K}_2^g]_{13} & [\mathbf{K}_2^g]_{14} \\ 0 & 0 & [\mathbf{K}_2^g]_{21} & [\mathbf{K}_2^g]_{22} & [\mathbf{K}_2^g]_{23} & [\mathbf{K}_2^g]_{24} \\ 0 & 0 & [\mathbf{K}_2^g]_{31} & [\mathbf{K}_2^g]_{32} & [\mathbf{K}_2^g]_{33} & [\mathbf{K}_2^g]_{34} \\ 0 & 0 & [\mathbf{K}_2^g]_{41} & [\mathbf{K}_2^g]_{42} & [\mathbf{K}_2^g]_{43} & [\mathbf{K}_2^g]_{44} \end{bmatrix}$$

Summarizing

In summary to solve a 2D truss problem, one needs to:

- establish a global numbering of the nodes;
- decide the orientation of each element;
- for each find the stiffness matrix in global coordinates;
- assemble the overall stiffness matrix considering the global numbering of dofs.

Costruction of the stiffness matrix in python

Recall the data for the problem

- $E = 210$ [GPa] (steel) for each element;
- $A = 4 \times 10^{-4}$ [m²], $A_1 = \sqrt{2}A$, $A_2 = A$;
- $F = 50$ [kN].
- $L = 1$ [m].

The stiffness matrix is obtained by looping over the elements, calling the function that construct the stiffness matrix in the global reference frame and assembling it in the overall stiffness matrix.

This procedure is illustrated by the following code:

```
In [71]: E = 210 * 1e9
A = 4 * 1e-4

A_1 = np.sqrt(2) * A
A_2 = A

array_EA = np.array([E*A_1, E*A_2])

F = 50 * 1e3

n_elements = connectivity_table.shape[0]
n_nodes = coordinates.shape[0]
n_dofs = 2 * n_nodes
K = np.zeros((n_dofs, n_dofs))

theta_vec = np.zeros((n_elements, ))

for ii in range(n_elements):

    left_node, right_node = connectivity_table[ii]

    K_element, theta_element = truss_2d_element(coordinates[left_node],
                                                coordinates[right_node],
                                                array_EA[ii])

    theta_vec[ii] = theta_element

    dof_left = 2*left_node
    dof_right = 2*right_node

    K[dof_left:dof_left+2, dof_left:dof_left+2] += K_element[0:2,0:2]
    K[dof_right:dof_right+2, dof_right:dof_right+2] += K_element[2:4,2:4]

    K[dof_left:dof_left+2, dof_right:dof_right+2] += K_element[0:2,2:4]
    K[dof_right:dof_right+2, dof_left:dof_left+2] += K_element[2:4,0:2]

print(f"Stiffness matrix normalized by axial stiffness: \n{K/(E*A/(L))}\n")
```

Stiffness matrix normalized by axial stiffness:

```
[[ 0.5  0.5 -0.5 -0.5  0.   0. ]
 [ 0.5  0.5 -0.5 -0.5  0.   0. ]
 [-0.5 -0.5  0.5  0.5  0.   0. ]
 [-0.5 -0.5  0.5  1.5  0.  -1. ]
 [ 0.   0.   0.   0.   0.   0. ]
 [ 0.   0.   0.  -1.   0.   1. ]]
```

Solving the FEM problem in python

To solve the FEM problem the external forces and the boundary conditions are applied. The solution is obtained by solving the linear system of equations after eliminating the degrees of freedom that are fixed. In these example the nodes undergoing boundary conditions are the nodes 1 and 3. Therefore the dofs 1,2,5,6 are eliminated.

So the Stiffness matrix without the constrained nodes is

$$\mathbf{K}_{\text{red}} = \begin{bmatrix} [\mathbf{K}]_{33} & [\mathbf{K}]_{34} \\ [\mathbf{K}]_{43} & [\mathbf{K}]_{44} \end{bmatrix}$$

The corresponding force vector is simply given by $\mathbf{f}_{\text{red}} = (F \ 0)^T$

```
In [72]: nodes_bcs = [0, 2]
dofs_bcs = [2*nodes_bcs[i] for i in range(len(nodes_bcs))] \
            + [2*nodes_bcs[i]+1 for i in range(len(nodes_bcs))]
dofs_bcs.sort()
dofs = np.arange(n_dofs)

dofs_no_bcs = list(set(dofs) - set(dofs_bcs))
K_red = K[np.ix_(dofs_no_bcs, dofs_no_bcs)]
f_red = np.array([F, 0])
q_red = np.linalg.solve(K_red, f_red)

print(f"Normalized displacement at node 2: \n {q_red/(F*L/(E*A))} \n")
```

Normalized displacement at node 2:
[3. -1.]

Postprocessing: computation of the reaction forces, normal stresses and forces

It is sufficient to multiply the stiffness matrix by the displacement to obtain the overall forces. The reaction forces are the forces at constrained nodes

```
In [73]: q = np.zeros((n_dofs, ))
q[dofs_no_bcs] = q_red
f = K @ q

for node in nodes_bcs:
    reaction = f[2*node:2*node+2]
    print(f"Normalized reaction at node {node + 1}: \n {reaction/F} \n")
```


Normalized reaction at node 1:

[-1. -1.]

Normalized reaction at node 3:

[0. 1.]

The strain in each element can be computed via the following formula

$$\varepsilon_x(x^l) = \frac{du^l}{dx^l} = \left(\frac{dx^l}{d\xi} \right)^{-1} \frac{du^l}{d\xi}$$

where again ξ is the reference coordinate. The first contribution is

$$\begin{aligned} \frac{dx^l}{d\xi} &= \begin{bmatrix} \frac{d}{d\xi} N_1 & \frac{d}{d\xi} N_2 \end{bmatrix} \begin{pmatrix} x_1^l \\ x_2^l \end{pmatrix} \\ &= \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{pmatrix} x_1^g \cos \theta_e + y_1^g \sin \theta_e \\ x_2^g \cos \theta_e + y_2^g \sin \theta_e \end{pmatrix} \end{aligned}$$

Similarly for the second contribution

$$\begin{aligned} \frac{du^l}{d\xi} &= \begin{bmatrix} \frac{d}{d\xi} N_1 & \frac{d}{d\xi} N_2 \end{bmatrix} \begin{pmatrix} u_1^l \\ u_2^l \end{pmatrix} \\ &= \begin{bmatrix} -1 & 1 \end{bmatrix} \begin{pmatrix} u_1^g \cos \theta_e + v_1^g \sin \theta_e \\ u_2^g \cos \theta_e + v_2^g \sin \theta_e \end{pmatrix} \end{aligned}$$

So the strain in each element is given by

$$\varepsilon_x(x^l) = \frac{u_2^l - u_1^l}{x_2^l - x_1^l} = \frac{(u_2^g - u_1^g) \cos \theta_e + (v_2^g - v_1^g) \sin \theta_e}{(x_2^g - x_1^g) \cos \theta_e + (y_2^g - y_1^g) \sin \theta_e}$$

```
In [74]: strain_vec = np.zeros((n_elements, ))
stress_vec = np.zeros((n_elements, ))
for ii in range(n_elements):

    left_node, right_node = connectivity_table[ii]
    x_glo_1, y_glo_1 = coordinates[left_node]
    x_glo_2, y_glo_2 = coordinates[right_node]

    u_glo_1, v_glo_1 = q[2*left_node:2*left_node+2]
    u_glo_2, v_glo_2 = q[2*right_node:2*right_node+2]

    strain_vec[ii] = ((u_glo_2 - u_glo_1)*np.cos(theta_vec[ii]) + (v_glo_2 -
                                                                /((x_glo_2 - x_glo_1)*np.cos(theta_vec[ii]) + (y_glo_2 -

stress_vec = array_EA * strain_vec
```

```
print(f"Normalized stress inside each element: \n {stress_vec/F} \n ")
```

Normalized stress inside each element:

```
[ 1.41421356 -1.          ]
```

Then element one undergoes traction whereas element 2 undergoes compression.

Resources

Check the LMS to obtain the PDF version of the notebook.

[Lecture notes from Duke University]

(<https://people.duke.edu/~hpgavin/cee421/truss-method.pdf>)