University of Nottingham
Department of Mechanical, Materials and Manufacturing Engineering

Computer Modelling Techniques

FE-02-01

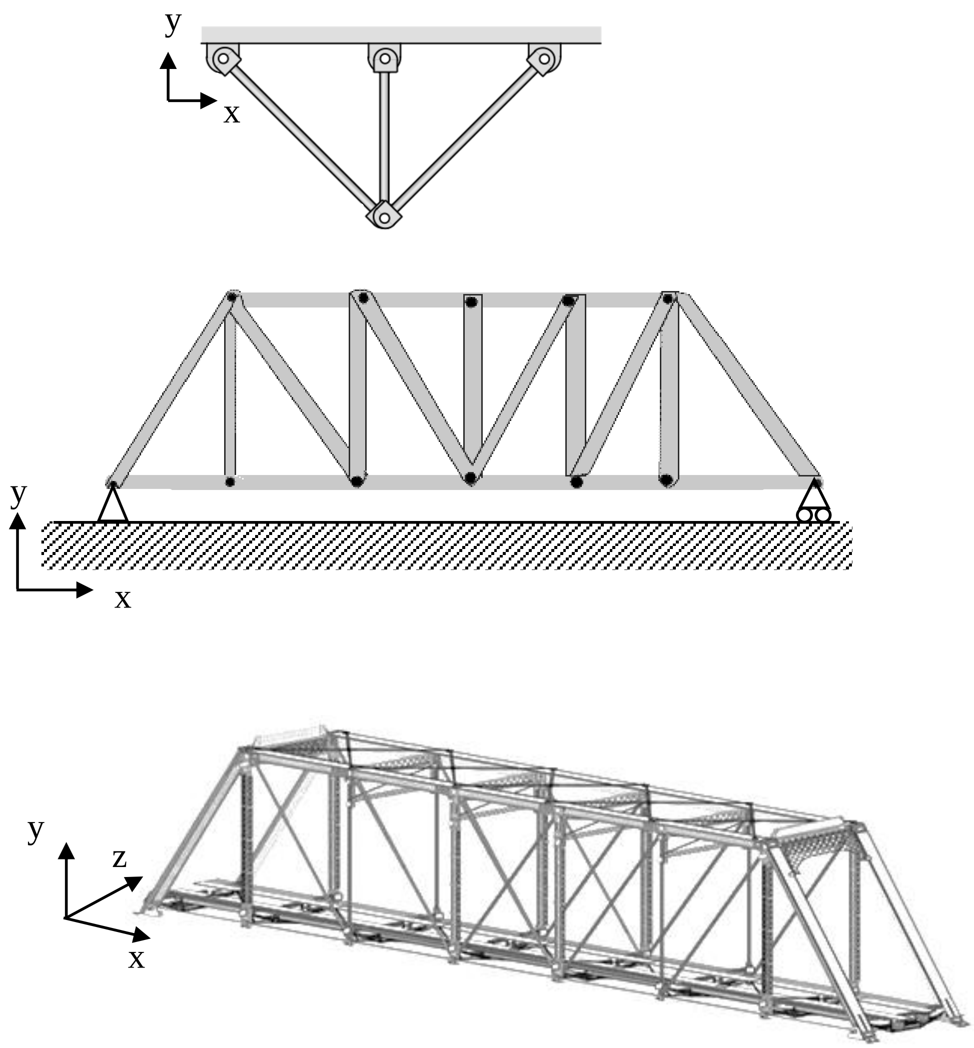
2D PIN-JOINTED FINITE ELEMENTS

Lecture Outline

2.1 Introduction

- The FE formulation is extended to cover two-dimensional (2D) pin-jointed truss structures in which the elements can be positioned at any angle.

- Therefore, each node will have two displacement components, u_x and u_y, i.e. there are two 'degrees of freedom' per node.



Examples of 2D and 3D pin-jointed structures

2.2 FE Formulation Steps

Step 1: Define the element and the shape functions

Assume linear, quadratic or higher-order interpolation functions for the displacements.

⇓

Step 2: Satisfy the material law (the constitutive equations)

Differentiate the displacement functions to obtain the strains, then use the material law (stress-strains relationships)

⇓

Step 3 : Derive the element stiffness matrix

Use either a direct equilibrium or an energy approach to obtain: $[k]_{element} [u]_{element} = [F]_{external}$

⇓

Step 4: Assemble the overall stiffness matrix

Assemble all elements together to form the final equations: $[K]_{global} [u]_{global} = [F]_{global}$

⇓

Step 5: Apply the boundary conditions and external loads

Satisfy the prescribed displacement and applied force conditions

⇓

Step 6: Solve the equations

Solve the system of linear algebraic equations to obtain the nodal displacements

⇓

Step 7: Compute other variables

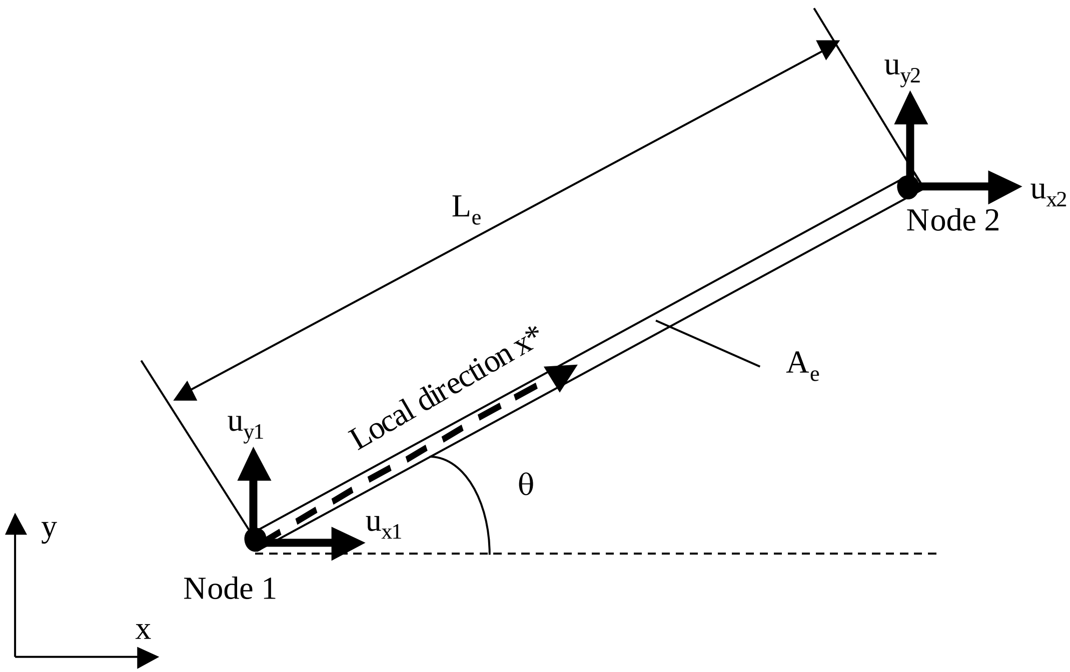
Strains can be obtained by differentiating the computed displacements over each element. Stresses can be obtained from the stress-strain relationships (constitutive equations)

Summary of the derivation of FE formulation

2.3 FE Formulation for 2D Pin-jointed Elements

Step 1: Define the element and the shape (interpolation) functions

The first step in any FE analysis is to divide the structure into elements and examine the behaviour of a typical element.



A 2-node Truss Element

- For pin-jointed element, it is convenient to define a *'local' axis*, i.e. an axis that is measured along the uniaxial direction of the element. The local (uniaxial) coordinate, x^*, is defined along the element.

- The element is assumed to have two degrees of freedom per node, the global (Cartesian) displacements in the x and y-direction, u_x and u_y, which are components of the local uniaxial displacement, u^*, along the element (in the direction of x^*).

- F_x and F_y are the global forces in the x and y-directions, which are components of the local uniaxial force, F^*, along the element (in the direction of x^*).

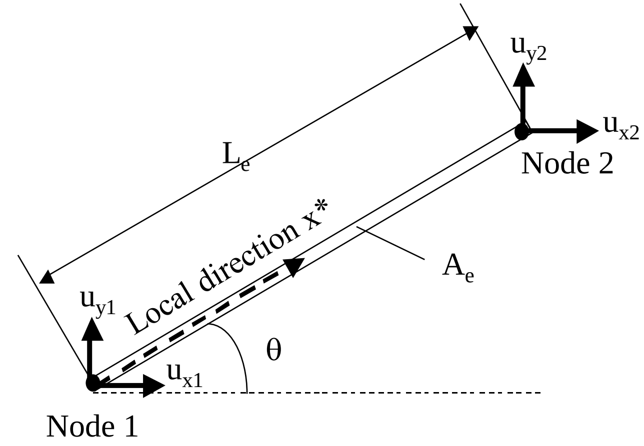
- For simplicity, the displacements, which are the unknown variables, will be assumed to vary linearly over each element, i.e. a constant strain (and stress) within each element, as follows:

$$u^* = C_1 + C_2 x^*$$

where C_1 and C_2 are constants.

The displacement conditions at the two nodes are:

- At node 1 (where $x^* = 0$), $u^* = u^*_1$
- At node 2 (where $x^* = L_e$), $u^* = u^*_2$



Therefore, the displacements of the two nodes can be written in terms of C_1 and C_2 in matrix form as follows:

$$\begin{bmatrix} u_1^* \\ u_2^* \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 1 & L_e \end{bmatrix} \begin{bmatrix} C_1 \\ C_2 \end{bmatrix}$$

These equations can be generalised as follows:

$$\lfloor u_e^* \rfloor = [A][C]$$

where

$[u^*_e]$ is the displacement vector of the element,

$[A]$ is a "*coordinate matrix*“

$[C]$ contains the constants C_1 and C_2

The element described here has a linear ‘*shape function*’.

In order to determine the constants C_1 and C_2, the matrix $[C]$ is moved to the left hand side, as follows:

$$[C]=[A]^{-1}\lfloor u_e^* \rfloor$$

Although it is convenient to use the displacement vector u^* along the element in the derivations, the global (Cartesian) x and y components of the displacement vector are often used in practice.

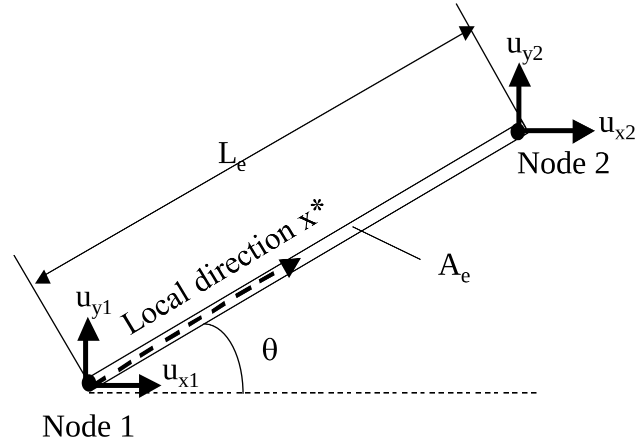
It is also convenient to group relevant variables together as matrices.

$$[u_e]=\begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \end{bmatrix} \;;\; [F_e]=\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \end{bmatrix}$$

where
$[u_e]$ is the Cartesian element displacement vector
$[F_e]$ is the Cartesian element force vector.

Note the order of the components in the vectors.

By resolving the displacements along the direction of the element, the global displacement components can be easily determined as follows:



$$u_1^* = u_{x1}\cos\theta + u_{y1}\sin\theta$$
$$u_2^* = u_{x2}\cos\theta + u_{y2}\sin\theta$$

Substituting these expressions results in :

$$[C] = \begin{bmatrix} C_1 \\ C_2 \end{bmatrix} = \begin{bmatrix} cos\theta & \sin\theta & 0 & 0 \\ \dfrac{-\cos\theta}{L_e} & \dfrac{-\sin\theta}{L_e} & \dfrac{\cos\theta}{L_e} & \dfrac{\sin\theta}{L_e} \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \end{bmatrix}$$

Step 2: Satisfy the material law (constitutive equations)

Since the displacement variable u^* is along the element, only one strain component exists, which can be easily determined by differentiating equation (1), as follows:

$$[\varepsilon] = \frac{du^*}{dx^*} = \frac{d}{dx^*}\left(C_1 + C_2\, x^*\right) = C_2$$

Note that the strain per element is therefore constant. Substituting C_2 gives the following expression for strain in terms of the displacement vector $[u_e]$:

$$[\varepsilon] = \frac{1}{L_e}\left[-\cos\theta \quad -\sin\theta \quad \cos\theta \quad \sin\theta\right]\begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \end{bmatrix}$$

or, in general:

$$[\varepsilon] = [B][u_e]$$

where $[B]$ is a "*dimension matrix*", defined as follows:

$$[B] = \frac{1}{L_e}\left[-\cos\theta \quad -\sin\theta \quad \cos\theta \quad \sin\theta\right]$$

The generalised Hooke's law expressions for strain-stress can be written in matrix form as follows:

$$[\sigma] = [D][\varepsilon]$$

where [D] is a "*material property matrix*".

In the pin-jointed element, [D]=E, since the load along the element is uniaxial.

Stresses can now be expressed as a function of displacements

$$[\sigma] = [D][B][u_e]$$

Step 3 : Derive the element stiffness matrix

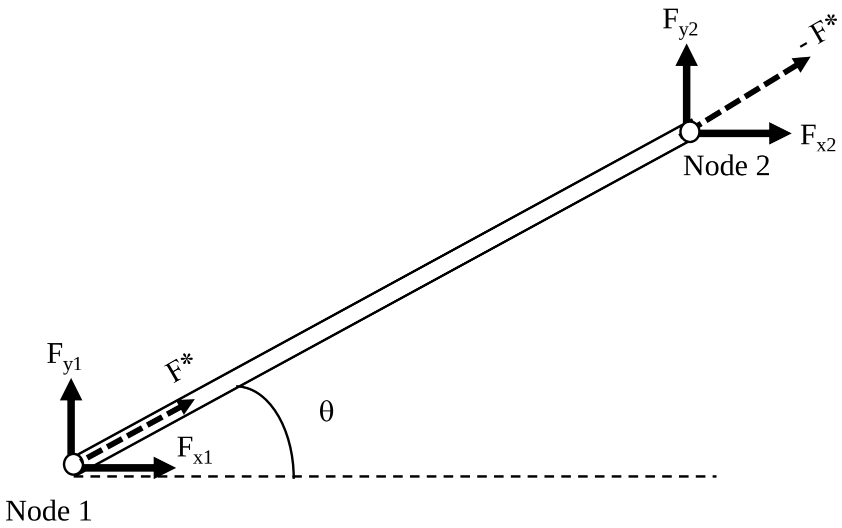
(a) Direct equilibrium approach

To satisfy equilibrium at the nodes, the forces at each node can be written as follows - see Figure

$$F_{x1} = -F^* \cos\theta$$
$$F_{y1} = -F^* \sin\theta$$
$$F_{x2} = F^* \cos\theta$$
$$F_{y2} = F^* \sin\theta$$

by using the expression for the matrix $[B]$, these force expressions can be expressed in matrix form as follows:

$$[F_e] = L_e [B]^T F^*$$



Equilibrium of the nodal points

The element stress can be simply obtained by dividing the uniaxial force F^* by the element area, i.e.

$$F^* = A_e [\sigma]$$

We can write an expression for the element forces in terms of displacements as follows:

$$[F_e] = A_e \; L_e \; [B]^T \; [D] \, [B] \; [u_e]$$

Therefore the element stiffness matrix $[k_e]$ can be defined as:

$$[F_e] = [k_e][u_e]$$

where $[k_e]$ is

$$[k_e] = A_e \; L_e \; [B]^T \; [D][B]$$

By substituting for $[B]$ and $[D]$, and further manipulation, the element stiffness matrix can be expressed as follows:

$$[k_e] = \left(\frac{A_e E}{L_e}\right) \begin{bmatrix} \cos^2\theta & \cos\theta\sin\theta & -\cos^2\theta & -\cos\theta\sin\theta \\ \cos\theta\sin\theta & \sin^2\theta & -\cos\theta\sin\theta & -\sin^2\theta \\ -\cos^2\theta & -\cos\theta\sin\theta & \cos^2\theta & \cos\theta\sin\theta \\ -\cos\theta\sin\theta & -\sin^2\theta & \cos\theta\sin\theta & \sin^2\theta \end{bmatrix}$$

Note that the element stiffness matrix is a symmetric matrix.

(b) Energy method

The total potential energy (TPE) of the element can be expressed as follows:

$$T.P.E. = \int_v \frac{1}{2} [\sigma]^T [\varepsilon]\, dv - [u_e]^T\ [F_e]$$

where v is the volume (here $dv = A_e\, dx$).

Substituting for the stress and strain results in:

$$\begin{aligned} T.P.E. &= \int_v \frac{1}{2} \left([D][B][u_e] \right)^T \left([B][u_e] \right) \left(A_e\, dx \right) - [u_e]^T [F_e] \\ &= \int_{L_e} \frac{1}{2} A_e [u_e]^T\ [B]^T\ [D][B]\ [u_e]\ dx - [u_e]^T [F_e] \\ &= \frac{1}{2} A_e [u_e]^T\ [B]^T\ [D][B]\ [u_e]\ L_e - [u_e]^T [F_e] \end{aligned}$$

Note that $[D]^T = [D]$ since $[D] = E$ is effectively symmetric.

Using the **principle of minimum total potential energy**, the differential of T.P.E. with respect to the displacement [*u*] must be zero.

It is convenient to present the differentiation of the TPE in a symbolic manner by treating [*u*] as if it were an algebraic variable, as follows:

$$\frac{\partial\left(\frac{1}{2}[u_e]^T\ [u_e].\right)}{\partial[u]} = \frac{\partial\left(\frac{1}{2}[u_e]^2\right)}{\partial[u]} = \frac{1}{2}\times 2\ [u_e] = [u_e]$$

Therefore, the differential of the TPE can be expressed in matrix form as follows:

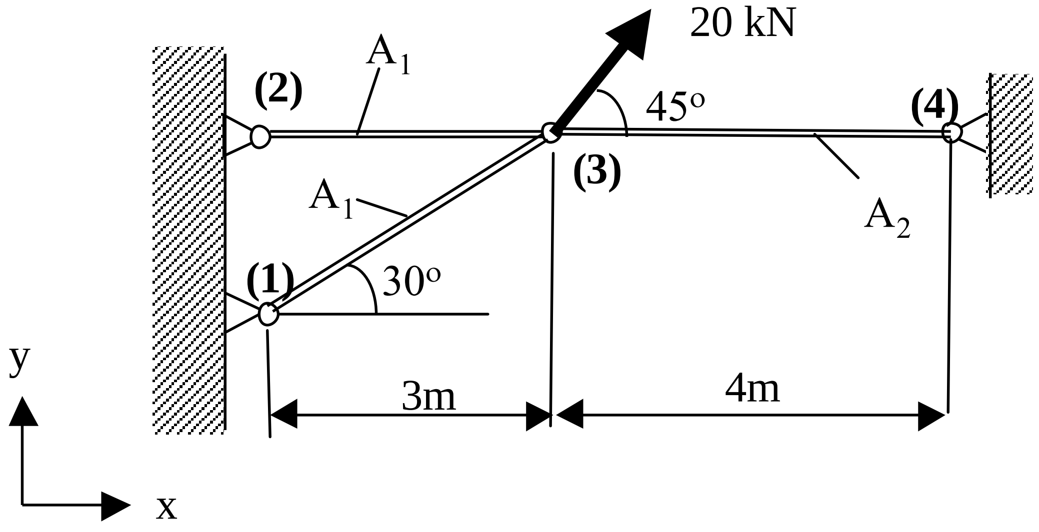
$$\frac{\partial(T.P.E.)}{\partial[u]} = 0 = \left(A_e\, L_e\ [B]^T[D][B]\right)\ [u_e]\ \text{-}\ [F_e]$$

which leads to the following expression for the element stiffness:

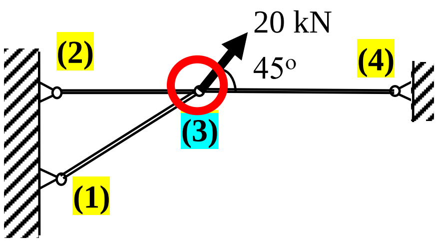
$$0 = [k_e][u_e] - [F_e]$$

which is identical to stiffness matrix derived using the direct equilibrium approach.

3.4 A Pin-jointed Structural Assembly Example



Structural analysis example



Problem Definition

(a) Geometry:

A frictionless pin-jointed structure consists of three members with the three lengths. The members have a cross-sectional area of A_1 (250 mm^2) or A_2 (450 mm^2)

(b) Material Properties:

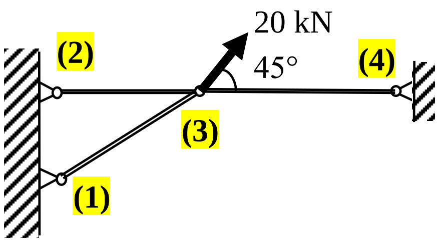
Young's modulus for all members is 207 GN/m^2.

(c) Boundary Conditions:

Points 1, 2 and 4 are fixed to a rigid surface, and a point force of 20 kN is applied to point 3 at an angle of 45°.

(d) Objective of the analysis:

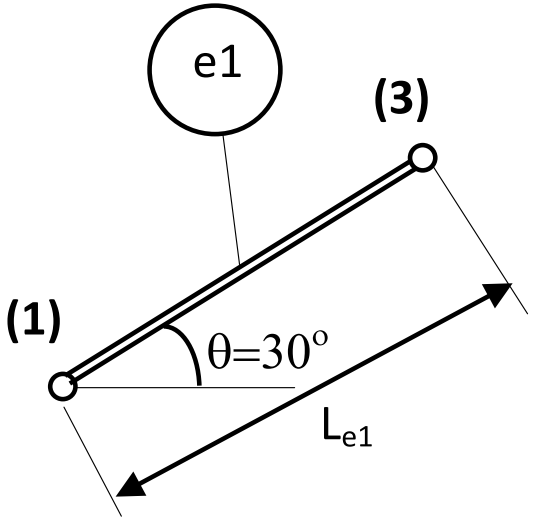
The objective is to calculate the **horizontal** and **vertical** components of the **displacement** at **point 3** and, using the calculated displacements, to determine the **stress** in the inclined member.



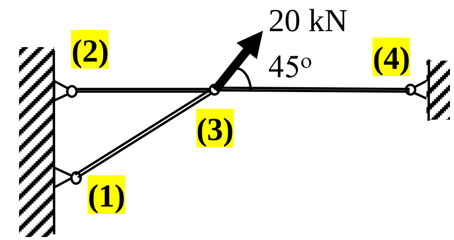
For element e1 :

First Node = Node 1
Second Node = Node 3
Angle θ = 30°
Length l_e = 3.464 m

Note that if the first and second nodes are swapped, the angle θ will be 210°. However, the element stiffness matrix remains the same whether θ = 30° or 210° is used.



Configuration of element e1



The force-displacement equations for this individual element is

$$\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{x3} \\ F_{y3} \end{bmatrix}_{e1} = 10^6 \begin{bmatrix} 11.205 & 6.469 & -11.205 & -6.469 \\ 6.469 & 3.735 & -6.469 & -3.735 \\ -11.205 & -6.469 & 11.205 & 6.469 \\ -6.469 & -3.375 & 6.469 & 3.735 \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x3} \\ u_{y3} \end{bmatrix}_{e1}$$

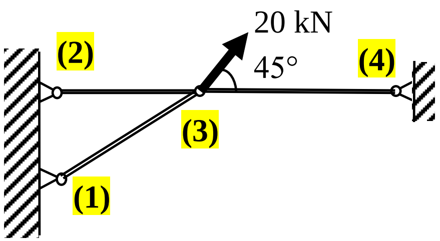
Putting this expression in the global system of equations results in the global stiffness matrix:

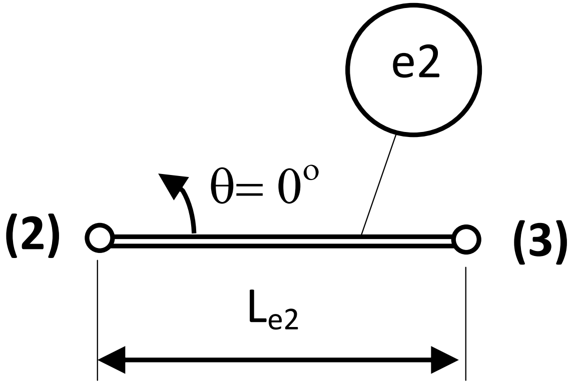
$$\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4} \end{bmatrix}_{e1} = 10^6 \begin{bmatrix} 11.205 & 6.469 & X & X & -11.205 & -6.469 & X & X \\ 6.469 & 3.735 & X & X & -6.469 & -3.735 & X & X \\ X & X & X & X & X & X & X & X \\ X & X & X & X & X & X & X & X \\ -11.205 & -6.469 & X & X & 11.205 & 6.469 & X & X \\ -6.469 & -3.735 & X & X & 6.469 & 3.735 & X & X \\ X & X & X & X & X & X & X & X \\ X & X & X & X & X & X & X & X \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \\ u_{x4} \\ u_{y4} \end{bmatrix}_{e1}$$

where X indicates no contribution to the $[K]$ matrix, i.e. a vacancy in the matrix.

For element e2 :

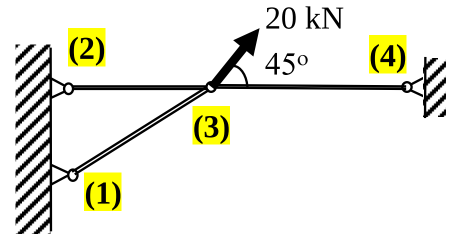
First Node = Node 2
Second Node = Node 3
Angle $\theta = 0°$
Length $l_e = 3$ m





Configuration of element e2

The force-displacement equations for this individual element is



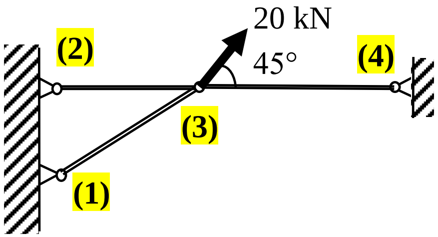
$$\begin{bmatrix} F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \end{bmatrix}_{e2} = 10^6 \begin{bmatrix} 17.25 & 0 & -17.25 & 0 \\ 0 & 0 & 0 & 0 \\ -17.25 & 0 & 17.25 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \end{bmatrix}_{e2}$$

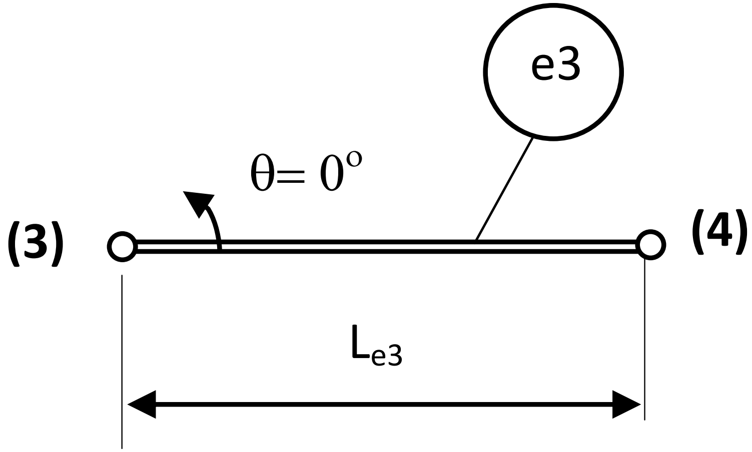
Putting this expression in the global system of equations results in the contribution

$$\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4} \end{bmatrix}_{e2} = 10^6 \begin{bmatrix} X & X & X & X & X & X & X & X \\ X & X & X & X & X & X & X & X \\ X & X & 17.25 & 0 & -17.25 & 0 & X & X \\ X & X & 0 & 0 & 0 & 0 & X & X \\ X & X & -17.25 & 0 & 17.25 & 0 & X & X \\ X & X & 0 & 0 & 0 & 0 & X & X \\ X & X & X & X & X & X & X & X \\ X & X & X & X & X & X & X & X \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \\ u_{x4} \\ u_{y4} \end{bmatrix}_{e2}$$

For element e3 :

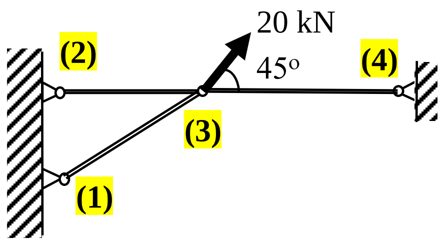
First Node = Node 3
Second Node = Node 4
Angle θ = 0°
Length l_e = 4 m





Configuration of element e3

The force-displacement equations for this individual element is



$$\begin{bmatrix} F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4} \end{bmatrix}_{e3} = 10^6 \begin{bmatrix} 23.29 & 0 & -23.29 & 0 \\ 0 & 0 & 0 & 0 \\ -23.29 & 0 & 23.29 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_{x3} \\ u_{y3} \\ u_{x4} \\ u_{y4} \end{bmatrix}_{e3}$$

Putting this expression in the global system of equations results in the contribution

$$\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4} \end{bmatrix}_{e3} = 10^6 \begin{bmatrix} X & X & X & X & X & X & X & X \\ X & X & X & X & X & X & X & X \\ X & X & X & X & X & X & X & X \\ X & X & X & X & X & X & X & X \\ X & X & X & X & 23.29 & 0 & -23.29 & 0 \\ X & X & X & X & 0 & 0 & 0 & 0 \\ X & X & X & X & -23.29 & 0 & 23.29 & 0 \\ X & X & X & X & 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \\ u_{x4} \\ u_{y4} \end{bmatrix}_{e3}$$

Step 4: Assemble the overall stiffness matrix

The next stage is to assemble all the individual elements together to form the overall structure (the assembly of the elements).

The displacement of a particular node must be the same for every element connected to it.

The externally applied forces at the nodes must also be balanced by the forces on the elements at these nodes, i.e.

$$[F]_{external} = \sum_{Aassembly} [F_e] = \sum_{Assembly} [k_e][u_e]$$

Therefore, a global system of equations can be written as follows:

$$[K]_{Assembly} \, [u]_{Assembly} = [F]_{Assembly}$$

where the matrices shown are the global assembly matrices containing all the nodal points.

For a problem with N nodes with two degrees of freedom per node:

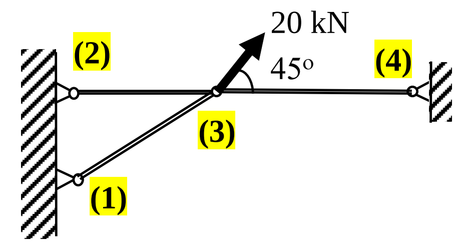
- The global $[K]_{Assembly}$ is of size $(2N \text{ x } 2N)$
- The global $[u]_{Assembly}$ and $[F]_{Assembly}$ vectors are of size $(2N \text{ x } 1)$

$$\underset{(2N\,x\,2N)}{\begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} & .. & .. & .. & K_{1,2N} \\ K_{21} & K_{22} & K_{23} & K_{24} & .. & .. & .. & K_{2,2N} \\ .. & .. & .. & .. & .. & .. & .. & .. \\ .. & .. & .. & .. & .. & .. & .. & .. \\ .. & .. & .. & .. & .. & .. & .. & .. \\ .. & .. & .. & .. & .. & .. & .. & .. \\ .. & .. & .. & .. & .. & .. & .. & .. \\ .. & .. & .. & .. & .. & .. & .. & K_{2N,2N} \end{bmatrix}_{Assembly}} \underset{(2N\,x1)}{\begin{bmatrix} u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ .. \\ .. \\ u_{xN} \\ u_{yN} \end{bmatrix}_{Assembly}} = \underset{(2N\,x1)}{\begin{bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ .. \\ .. \\ F_{xN} \\ F_{yN} \end{bmatrix}_{Assembly}}$$

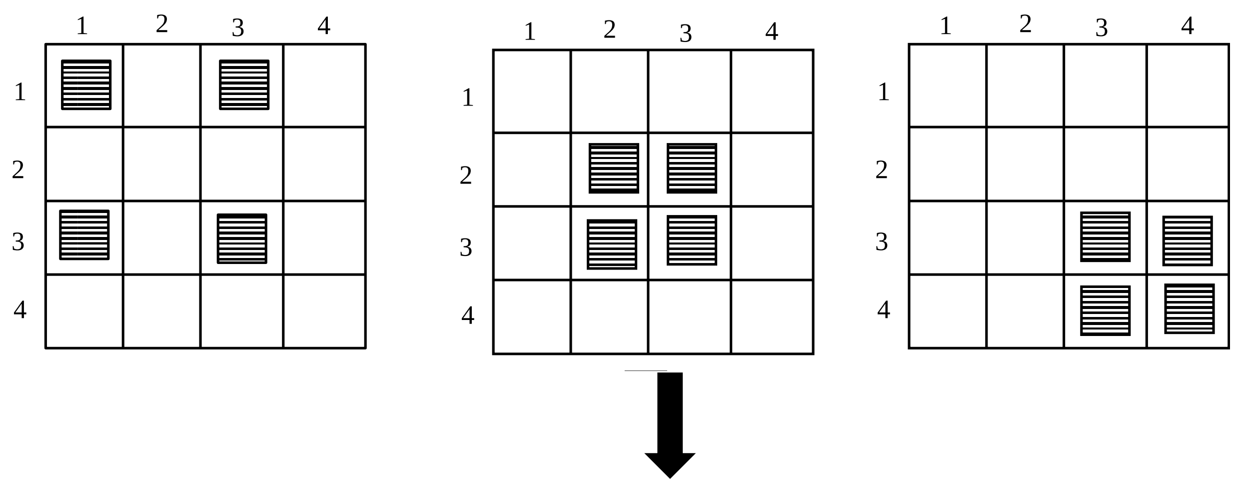
Note that the global stiffness matrix $[K]_{Assembly}$ is **sparsely populated** (i.e. containing relatively few non-zero coefficients), even in structures containing a large number of elements.

This is because not more than a few elements are connected to any one node.

The global force-displacement equations for the whole assembly can be obtained by combining the stiffness matrix contributions of all the individual elements such that the [K] coefficients belonging to common nodes are added together.

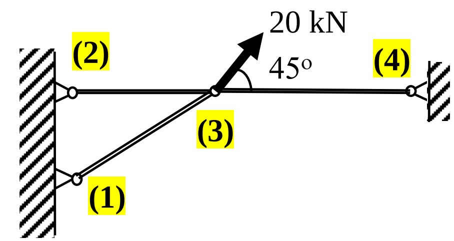


Element **e1** (nodes 1 and 3) | Element **e2** (nodes 2 and 3) | Element **e3** (nodes 3 and 4)



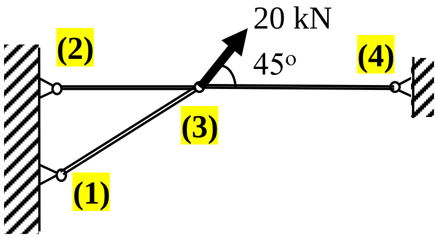
Assembled Global Stiffness Matrix

	1	2	3	4
1	e1	0	e1	0
2	0	e2	e2	0
3	e1	e2	e1 e2 e3	e3
4	0	0	e3	e3



The overall assembled global matrices are therefore:

$$
\left(10^6\right)
\begin{bmatrix}
11.205 & 6.469 & X & X & -11.205 & -6.469 & X & X \\
6.469 & 3.735 & X & X & -6.469 & -3.375 & X & X \\
X & X & 17.25 & 0 & -17.25 & 0 & X & X \\
X & X & 0 & 0 & 0 & 0 & X & X \\
-11.205 & -6.469 & -17.25 & 0 & \text{q } (11.25 + 17.25 + 23.29) & (6.469 + 0 + 0) & -23.29 & 0 \\
-6.469 & -3.375 & 0 & 0 & (6.469 + 0 + 0) & (3.735 + 0 + 0) & 0 & 0 \\
X & X & X & X & -23.29 & 0 & 23.29 & 0 \\
X & X & X & X & 0 & 0 & 0 & 0
\end{bmatrix}
\begin{bmatrix}
u_{x1} \\ u_{y1} \\ u_{x2} \\ u_{y2} \\ u_{x3} \\ u_{y3} \\ u_{x4} \\ u_{y4}
\end{bmatrix}
=
\begin{bmatrix}
F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4}
\end{bmatrix}
$$



Step 5: Apply the boundary conditions and external loads

The displacement restraints at points 1, 2 and 4 are :

$$u_{x1} = u_{y1} = 0$$

$$u_{x2} = u_{y2} = 0$$

$$u_{x4} = u_{y4} = 0$$

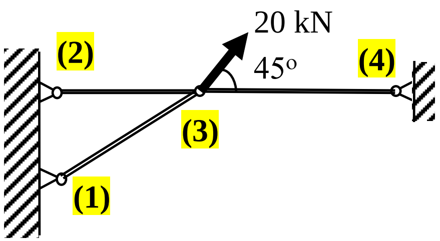
and the external force vector contains only two components at node 3, as follows:

$$F_{x3} = 20 \cos 45^o \times 10^3 \ N$$

$$F_{y3} = 20 \sin 45^o \times 10^3 \ N$$

Therefore the global $[u]_{Assembly}$ and $[F]_{Assembly}$ vectors can be written as follows:

$$[u]_{Assembly} = \begin{bmatrix} 0 \\ 0 \\ \hline 0 \\ 0 \\ \hline u_{x3} \\ u_{y3} \\ \hline 0 \\ 0 \end{bmatrix} ; \quad [F]_{Assembly} = \begin{bmatrix} 0 \\ 0 \\ \hline 0 \\ 0 \\ \hline F_{x3} \\ F_{y3} \\ \hline 0 \\ 0 \end{bmatrix}$$



Step 6: Solve the equations

Standard equation solvers, such as the Gaussian elimination technique can be used to solve the equations to determine the unknown variables (here only the displacements) at each node.

Since the stiffness matrix is symmetric and sparsely populated, specially adapted solvers that can considerably reduce the memory storage requirements are often used.

Since the displacements at nodes 1, 2 and 4 are already given, the rows and columns multiplying these displacements can be eliminated.

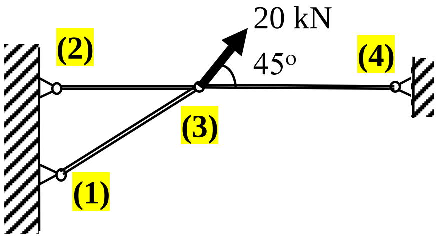
$$10^6 \begin{bmatrix} 51.745 & 6.469 \\ 6.469 & 3.735 \end{bmatrix} \begin{bmatrix} u_{x3} \\ u_{y3} \end{bmatrix} = 10^3 \begin{bmatrix} 20\cos 45^o \\ 20\sin 45^o \end{bmatrix}$$

from which the displacements of node 3 can be calculated as:

$$u_{x3} = -0.256\ mm$$

$$u_{y3} = 4.229\ mm$$

Note that a negative displacement indicates movement to the left.



Step 7: Compute other variables

Once the displacements of the nodal points are computed, other parameters such as element stresses can be calculated.

$$[\sigma_e] = [B][D][u_e]$$

Therefore, the stress in element e_1 can be calculated as follows:

$$[\sigma_{e1}] = \left(\frac{207\times10^9}{3.464}\right)\begin{bmatrix}-0.866 & -0.5 & 0.866 & 0.5\end{bmatrix}\begin{bmatrix}0\\0\\u_{x3}\\u_{y3}\end{bmatrix} = 113.1\times10^6\ N/m^2$$

Similarly, the stresses in elements e_2 and $e3$ can be calculated, i.e.

$$[\sigma_{e2}] = -17.6\ MN/m^2$$

$$[\sigma_{e3}] = 13.2\ MN/m^2$$

2.5 Summary of Key Points

- In deriving the FE formulation for 2D pin-jointed structures, the first step is to define the element geometry and the order of variation of the displacement over each element.

- For each element, an individual element stiffness expression is derived first and elements are then assembled together.

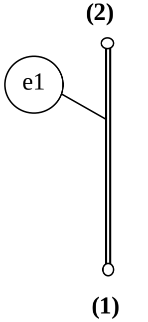
- In structural analysis problems, the individual element stiffness matrices are assembled together by superimposing the [K] coefficients of the nodes which are shared between two or more elements.

- The stiffness matrix of the assembly is always symmetric and sparsely populated even if a large number of elements is used in the FE mesh.

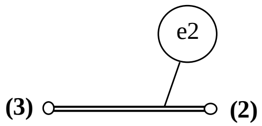
- The FE formulation for 2D pin-jointed elements can be easily extended, using the matrix expressions, to more sophisticated problems, e.g. truss elements in three-dimensional problems.

2.6 More Pin-jointed structural examples

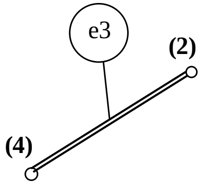
Pin-Jointed Structural Example-1



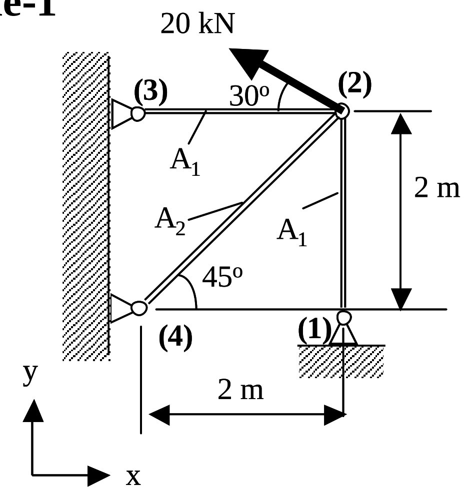
	1	2	3	4
1				
2				
3				
4				



	1	2	3	4
1				
2				
3				
4				



	1	2	3	4
1				
2				
3				
4				



Assembled stiffness matrix

	1	2	3	4
1				
2				
3				
4				