Matrix Structural Analysis – the Stiffness Method[*]

Matrix structural analyses solve practical problems of trusses, beams, and frames. The stiffness method is currently the most common matrix structural analysis technique because it is amenable to computer programming. It is important to understand how the method works. This document is essentially a brief introduction to the stiffness method (known as the finite element method, particularly when applied to continuum solid components).

Axial Bars (1-Dim)

For their simplicity, axial bars are useful in illustrating the method. We will show the basic data to be inputted to a computer program. Fig. 1 shows a 1-dim axially loaded bar. Let P = 24 kN, A_{ADC} = 400 mm^2, A_{CB} = 600 mm^2, L = 80 mm, and E = 200 GPa.

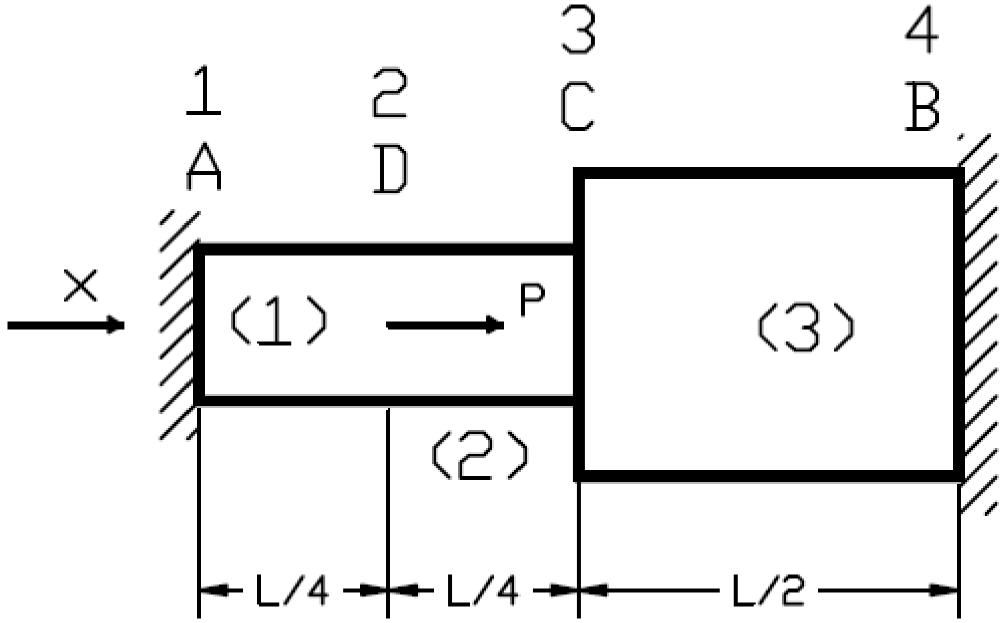


Fig. 1

A typical computer program should calculate the x-displacement u of all basic points (named nodes). The nodes of the bar are points A, D, C, and B. The displacements of nodes A and B are known in advance, simply each is equal to zero. Therefore, a computer program should calculate the displacements of nodes D and C (u_D and u_C). A program should calculate the reaction forces and the forces transmitted through the bar. Moreover, it should calculate the normal stresses at the segments AD, DC, and CB. Each segment is named an element.

[*] A. Mansour

Input Data

The coordinates of the nodes are given below:

Node number	Label of Fig.1	X coordinate - m
1	A	0.0
2	D	0.002
3	C	0.004
4	B	0.008

We should inform the program of the nodes associated with each element.

Element number	Label of Fig. 1	1st node	2nd node
1	AD	1	2
2	DC	2	3
3	CB	3	4

The previous two tables give the information required to calculate the length of each element. For instance, the length of element (2), $L_{(2)} = 0.004 - 0.002 = 0.002$ m. By the same token $L_{(3)} = 0.008 - 0.004 = 0.004$ m.

We should specify the material of each element or the relevant properties for each element.

Element number	Young's modulus (E) - Pa
1	200×10^9
2	200×10^9
3	200×10^9
4	200×10^9

<u>Displacement Boundary Conditions (B.C.)</u>
We know in advance that nodes 1 and 4 are fixed (since 1 and 4 are A and B).

Node number	u
1	0.0
4	0.0

<u>Force (load) Boundary Conditions</u>

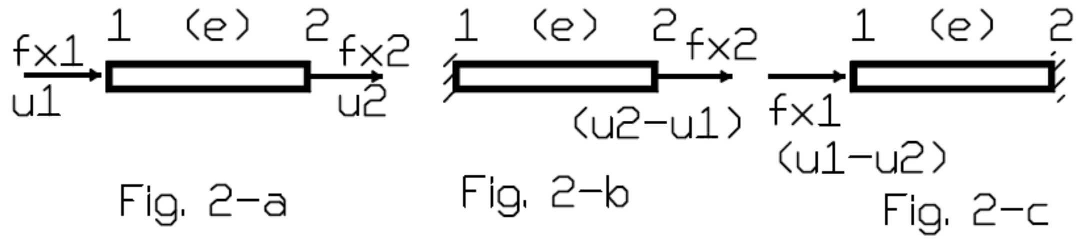
The forces at nodes D and C are known in advance. The following table gives these boundary conditions:

Node number	F_x - (N)
2	+24 000
3	0.0

F_{x2} is positive because it is in the positive x direction. Usually if u for any node is known in advance, then F for that node is unknown, and vice versa.

Having a full description of the problem, computer programs can determine all the nodal displacements and forces. The relationship among these variables is given below.

Stiffness Matrix



A typical element (e) is shown in Fig. 2a. The x-displacement of nodes 1 and 2 are u_1 and u_2. The nodal forces are f_{x1} and f_{x2}. Of course, $f_{x1} = -f_{x2}$. However, in order to have a systematic representation, we will keep a separate name for each nodal force.

The element is elastic and by consulting Fig. 2b,

$f_{x2} = k_{(e)}\ (u_2 - u_1) = k_{(e)}\ (-u_1 + u_2)$

Where, $k_{(e)} = EA / L$; the elemental stiffness.

Fig. 2c shows that

$f_{x1} = k_{(e)}\ (u_1 - u_2)$

Where, f_{x1} is a compressive force and $(u_1 - u_2)$ represents a corresponding contraction of the length of the element.

The following matrix equation represents the previous two equations.

$$\begin{pmatrix} f_{x1} \\ f_{x2} \end{pmatrix}_e = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}_e \begin{pmatrix} u_1 \\ u_2 \end{pmatrix} \qquad or \qquad (f)_e = [k]_e (u)$$

Where $[\ k\ ]_e$ is a 2 x 2 stiffness matrix. Now we can see why the method is named matrix structural analysis or stiffness method.

Temperature Effect

We need to include the effect of temperature rise $\Delta T = T - T_0$. Fig. 2b gives:

$u_2 - u_1 = f_{x2} / k_{(e)} + \alpha\ L\ \Delta T$

In addition, Fig. 2c gives

$u_1 - u_2 = f_{x1} / k_{(e)} - \alpha\, L\, \Delta T$

where, ($u_1 - u_2$) implies that node 1 moves in the positive x direction (the right direction). On the other hand, $\alpha\, L\, \Delta T$ implies that node 1 moves to the left to allow for the increase in length due to ΔT. This explains why ($- \alpha\, L\, \Delta T$) must be used.

$$EA\alpha\,\Delta\,T\begin{pmatrix} -1 \\ 1 \end{pmatrix} + \begin{pmatrix} f_{x1} \\ f_{x2} \end{pmatrix}_e = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix}_e \begin{pmatrix} u_1 \\ u_2 \end{pmatrix}$$

Degrees of Freedom

Each node can move in the x direction only. Therefore, each node has only one degree of freedom. Computer programs would address the displacements by their degrees of freedom (DOF). The displacements of nodes 1, 2, 3 and 4 correspond to degrees of freedom 1 up to 4. In addition, f_{x1} up to f_{x4} corresponds to degrees of freedom 1 up to 4.

Basic Steps in the Method

We will explain the method through the example of Fig. 1. We will calculate the nodal forces and elemental forces for this bar.

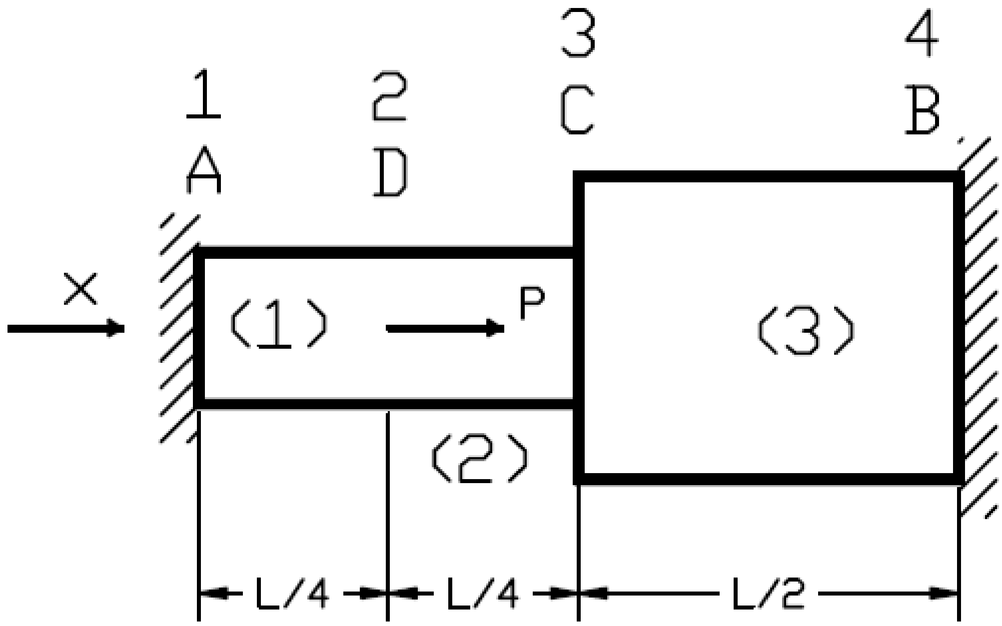


Fig. 1

The stiffness of each element is:

$k_1 = E_1 A_1 / L_1 =$	$(200 \times 10^9)\ (400 \times 10^6)\ /\ (0.020) =$	4.0×10^9 N/m,
$k_2 = E_2 A_2 / L_2 =$	$(200 \times 10^9)\ (400 \times 10^6)\ /\ (0.020) =$	4.0×10^9 N/m,
$k_3 = E_3 A_3 / L_3 =$	$(200 \times 10^9)\ (600 \times 10^6)\ /\ (0.040) =$	3.0×10^9 N/m.

For element 1:

$$\begin{pmatrix} f_{x1} \\ f_{x2} \end{pmatrix}_{(1)} = 10^9 \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix}_{(1)} \begin{pmatrix} u_1 \\ u_2 \end{pmatrix} \qquad or \qquad [k]_{(1)} = 10^9 \begin{array}{c} \begin{array}{cc} \mathit{1} & \mathit{2} \end{array} \quad \mathit{DOF} \\ \begin{bmatrix} (4) & (-4) \\ (-4) & (4) \end{bmatrix} \begin{array}{c} 1 \\ 2 \end{array} \end{array}$$

For identification purposes, the coefficients of the stiffness matrix of element 1 are surrounded by one set of round bracket (..). The coefficients for element 2 would be surrounded by two sets of brackets and so forth. This would help us to keep track of these coefficients in the subsequent steps. Moreover, the columns and rows of the matrix are identified by their corresponding DOF (1 and 2 for element 1). For instance, the coefficient in the **first** row and **second** column is $k_{12} = (-4) \times 10^9$ N/m

For element 2:

$$\begin{pmatrix} f_{x2} \\ f_{x3} \end{pmatrix}_{(2)} = 10^9 \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix}_{(2)} \begin{pmatrix} u_2 \\ u_3 \end{pmatrix} \qquad or \qquad [k]_{(2)} = 10^9 \begin{array}{c} \begin{array}{cc} \mathit{2} & \mathit{3} \end{array} \quad \mathit{DOF} \\ \begin{bmatrix} ((4)) & ((-4)) \\ ((-4)) & ((4)) \end{bmatrix} \begin{array}{c} 2 \\ 3 \end{array} \end{array}$$

For element 3:

$$\begin{pmatrix} f_{x3} \\ f_{x4} \end{pmatrix}_{(3)} = 10^9 \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix}_{(3)} \begin{pmatrix} u_3 \\ u_4 \end{pmatrix} \qquad or \qquad [k]_{(3)} = 10^9 \begin{array}{c} \begin{array}{cc} \mathit{3} & \mathit{4} \end{array} \quad \mathit{DOF} \\ \begin{bmatrix} (((4))) & (((-4))) \\ (((-4))) & (((4))) \end{bmatrix} \begin{array}{c} 3 \\ 4 \end{array} \end{array}$$

As mentioned above, the coefficients of the stiffness matrix of elements two and three are surrounded by two and three round brackets respectively.

We want to relate the nodal forces and displacements of the whole bar as follows:

$$\begin{pmatrix} F_{x1} \\ F_{x2} \\ F_{x3} \\ F_{x4} \end{pmatrix} = \begin{array}{c} \begin{array}{cccc} \mathit{1} & \mathit{2} & \mathit{3} & \mathit{4} \end{array} \ldots\ldots\ldots\ldots\ldots\ldots\ \mathit{DOF} \\ \begin{bmatrix} K_{11} & K_{12} & K_{13} & K_{14} \\ K_{21} & K_{22} & K_{23} & K_{24} \\ K_{31} & K_{32} & K_{33} & K_{34} \\ K_{41} & K_{42} & K_{43} & K_{44} \end{bmatrix}_{STRUCTURE} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{pmatrix} \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \end{array} \end{array}$$

Where the coefficients of the structure matrix K_{ij} are constructed from the coefficients of the individual stiffness matrices. We place each entry according to its associated DOF, as shown below:

$$
\begin{array}{ccc}
 & \begin{array}{cccc} \mathit{1} & \mathit{2} & \mathit{3} & \mathit{4} \;\ldots\ldots\ldots\ldots\ldots \end{array} & \mathit{DOF}\\
\begin{pmatrix}F_{x1}\\F_{x2}\\F_{x3}\\F_{x4}\end{pmatrix} = 10^9 & \begin{bmatrix} (4) & (-4) & & \\ (-4) & (4)+((4)) & ((-4)) & \\ & ((-4)) & ((4))+(((3))) & (((-3))) \\ & & (((-3))) & (((3))) \end{bmatrix}_{STRUCTURE} \begin{pmatrix}u_1\\u_2\\u_3\\u_4\end{pmatrix} & \begin{array}{c}1\\2\\3\\4\end{array}
\end{array}
$$

In the above structure stiffness matrix, empty entries show up because there is no element connecting nodes 1 and 3, 1 and 4, and 2 and 4. These entries must be replaced by zeroes as follows.

$$
\begin{pmatrix}F_{x1}\\F_{x2}\\F_{x3}\\F_{x4}\end{pmatrix} = 10^9 \begin{bmatrix} 4 & -4 & 0 & 0 \\ -4 & 8 & -4 & 0 \\ 0 & -4 & 7 & -3 \\ 0 & 0 & -3 & 3 \end{bmatrix}_{STRUCTURE} \begin{pmatrix}u_1\\u_2\\u_3\\u_4\end{pmatrix}
$$

<u>Solution of the System of Equations</u>

The above matrix equation corresponds to 4 equations. The unknowns are u_2, u_3, F_{x1}, and F_{x4}.

Since $u_1 = u_4 = 0$, then the coefficients of the stiffness matrix in the first and fourth columns are always multiplied by zeroes. Hence, we ignore columns 1 and 4. In addition, equations 1 and 4 correspond to the unknown forces F_{x1} and F_{x4}. Thus, we can use these equations later to determine F_{x1} and F_{x4}. For the time being we are going to use a subset of the matrix, that does not contain the columns and rows 1 and 4, as shown below.

$$
\begin{pmatrix} F_{x2} = 24000 \\ F_{x3} = 0 \end{pmatrix} = 10^9 \begin{bmatrix} 8 & -4 \\ -4 & 7 \end{bmatrix} \begin{pmatrix} u_2 \\ u_3 \end{pmatrix}
$$

Solve these equations to get $u_2 = 4.2\ 10^{-6}$ m and $u_3 = 2.4\ 10^{-6}$ m.

Now, we get the forces from equations 1 and 4:

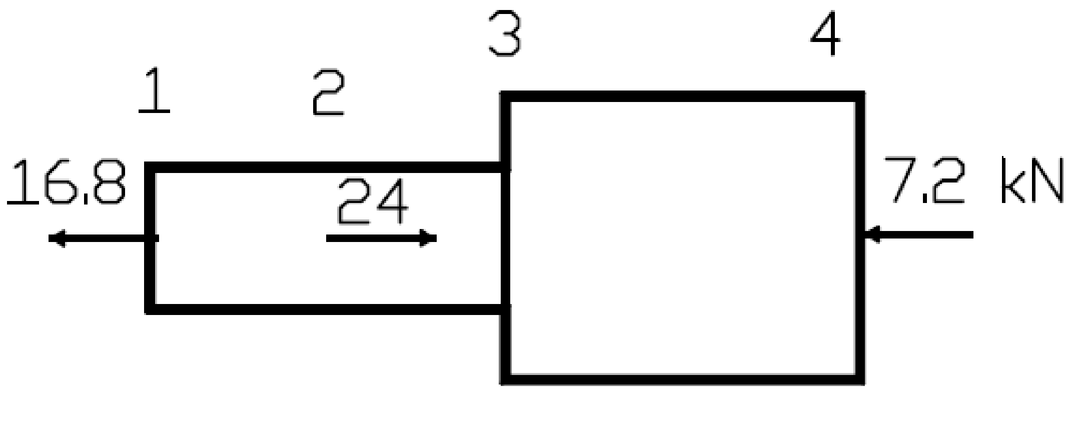


Fig. 3

$F_{x1} = (-4\ 10^9)\ u_2 + (\ 0.0\ )\ u_3 = -16800$ N, and
$F_{x4} = (\ 0.0\ )\ u_2 + (-3\ 10^9)\ u_3 = -7200$ N.

Fig. 3 shows the forces acting on the bar. The forces satisfy the equilibrium equation. The reaction forces are calculated correctly.

Determination of Forces at Each Element

We substitute the calculated displacements in the force-displacement matrix equation of each element.

Element 1

$$\begin{pmatrix} f_{x1} \\ f_{x2} \end{pmatrix} = 10^9 \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix} \begin{pmatrix} u_1 = 0 \\ u_2 = 4.2\ 10^{-6} \end{pmatrix} = \begin{pmatrix} -16800 \\ 16800 \end{pmatrix} \quad N$$

Element 2

$$\begin{pmatrix} f_{x2} \\ f_{x3} \end{pmatrix} = 10^9 \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix} \begin{pmatrix} u_2 = 4.2\ 10^{-6} \\ u_3 = 2.4\ 10^{-6} \end{pmatrix} = \begin{pmatrix} 7200 \\ -7200 \end{pmatrix} \quad N$$

and element 3

$$\begin{pmatrix} f_{x3} \\ f_{x4} \end{pmatrix} = 10^9 \begin{bmatrix} 3 & -3 \\ -3 & 3 \end{bmatrix} \begin{pmatrix} u_3 = 2.4\ 10^{-6} \\ u_4 = 0 \end{pmatrix} = \begin{pmatrix} 7200 \\ -7200 \end{pmatrix} \quad N$$

Fig. 4 shows that the forces acting on each element are indeed in equilibrium. The external forces at any node also must be in equilibrium with the forces transmitted to the bar. Fig. 4 shows the equilibrium of node 2. We can see that the external force F_{x2} = 24.0 kN is in equilibrium with the elemental (internal) forces ($f_{x2\,(1)} + f_{x2\,(2)}$ = 16.8 + 7.2 = 24.0 kN).

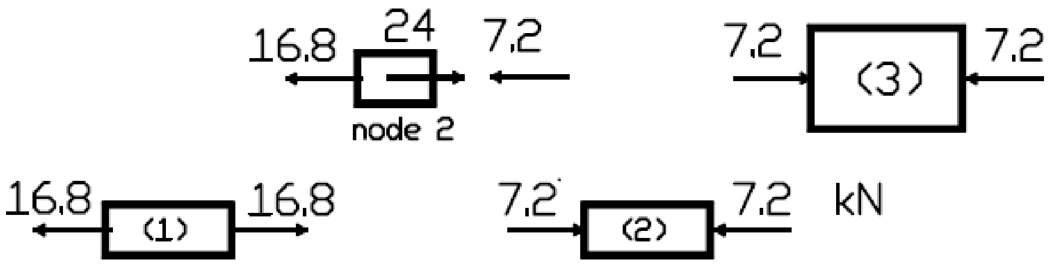


Fig. 4

<u>Stresses in Each Element</u>

We calculate the stresses in each element by dividing the elemental force by the area of the cross section.
$\sigma_{x\,(1)} = f_{x1} / A_{(1)} = 16800 / 400\ 10^{-6} = 42\ 10^6$ Pa = 42 MPa (T)
$\sigma_{x\,(2)} = f_{x2} / A_{(2)} = (-7200) / 400\ 10^{-6} = -18\ 10^6$ Pa = -18 MPa (C)
$\sigma_{x\,(3)} = f_{x3} / A_{(3)} = (-7200) / 600\ 10^{-6} = -12\ 10^6$ Pa = -12 MPa (C)

Strains of Each Element

$\varepsilon_{x(1)} = \sigma_{x(1)} / E_{(1)} = 42\ 10^{6} = 0.00021 = 0.21\ 10^{-3}$,
$\varepsilon_{x(2)} = \sigma_{x(2)} / E_{(2)} = -90.0\ 10^{-6}$, and $\varepsilon_{x(3)} = -60.0\ 10^{-6}$.

Alternatively, we can calculate the strains from nodal displacements,

$\varepsilon_{x(1)} = (u_2 - u_1) / L_1 = (4.2\ 10^{-6} - 0) / 0.020 = 0.21\ 10^{-3}$ and so on.

Example (1)

.

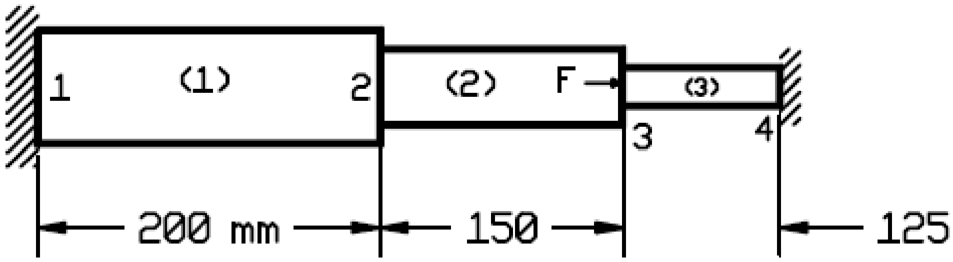


Fig. 5

The bar of Fig. 5 is subjected to F_3 = 15 kN, and ΔT = 20°C. $A_{(1)}$= 100 mm^2, $A_{(2)}$ = 75, $A_{(3)}$ = 50, E = 200 GPa, and $\alpha = 12 \times 10^{-6}$ / °C
Determine the nodal displacements, the reactions, and the forces transmitted through each element. Compare between the developed states of stresses for ΔT = 20°C and ΔT = 0°C

Solution
The stiffness of each element is:

$k_1 = (200 \times 10^9)(100 \times 10^{-6}) / 0.2 = 100 \times 10^6$ N/m
$k_2 = 100 \times 10^6$ N/m
$k_3 = 80 \times 10^6$ N/m

We should calculate the thermal terms ($\alpha E A \Delta T$) for each element:

$(\alpha E A \Delta T)_{(1)}$ = 4800 N
$(\alpha E A \Delta T)_{(2)}$ = 3600 N
$(\alpha E A \Delta T)_{(3)}$ = 2400 N

The element equations are:

$$\begin{pmatrix} f_{x1} \\ f_{x2} \end{pmatrix}_{(1)} + \begin{pmatrix} -4800 \\ 4800 \end{pmatrix} = 10^6 \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \begin{pmatrix} u_1 \\ u_2 \end{pmatrix} \quad ;$$

$$\begin{pmatrix} f_{x2} \\ f_{x3} \end{pmatrix}_{(2)} + \begin{pmatrix} -3600 \\ 3600 \end{pmatrix} = 10^6 \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \begin{pmatrix} u_2 \\ u_3 \end{pmatrix} \quad ;$$

$$\begin{pmatrix} f_{x3} \\ f_{x4} \end{pmatrix}_{(3)} + \begin{pmatrix} -2400 \\ 2400 \end{pmatrix} = 10^6 \begin{bmatrix} 100 & -100 \\ -100 & 100 \end{bmatrix} \begin{pmatrix} u_3 \\ u_4 \end{pmatrix}$$

Assemble the equations:

$$\begin{pmatrix} F_{x1} \\ F_{x2} \\ F_{x3} \\ F_{x4} \end{pmatrix} + \begin{pmatrix} (-4800) \\ (4800) + ((-3600)) \\ ((3600)) + (((-2400))) \\ (((2400))) \end{pmatrix} = 10^6 \begin{bmatrix} 100 & -100 & 0 & 0 \\ -100 & 200 & -100 & 0 \\ 0 & -100 & 180 & -80 \\ 0 & 0 & -80 & 80 \end{bmatrix} \begin{pmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{pmatrix}$$

Boundary conditions

$u_1 = 0$, $u_4 = 0$ ……(we may ignore rows and columns 1 and 4)

$F_{x2} = 0$, $F_{x3} = 15000$. Hence,

$$\begin{pmatrix} 1200 \\ 16200 \end{pmatrix} = 10^6 \begin{bmatrix} 200 & -100 \\ -100 & 180 \end{bmatrix} \begin{pmatrix} u_2 \\ u_3 \end{pmatrix}$$

(The above stiffness equation is the reduced stiffness matrix after applying the boundary conditions.)

Solve the equations to get

u_2 = 0.000070615	m
u_3 = 0.00012923	m

The reactions are obtained from rows 1 and 4. The following table shows the reactions at the support when ΔT = 20°C as well as when ΔT = 0°C.

	ΔT = 20°C	ΔT = 0°C
F_{x1} kN	-2.2615	-5.76923
F_{x4} kN	-12.7384	-9.23072

We can use the element matrix equations to get the forces acting on each element.

Element (1)	Element (2)	Element (3)
f_{x1} = -2.2615 kN	f_{x2} = -2.2615	f_{x3} = 12.7
f_{x2} = 2.2615	f_{x3} = 2.2615	f_{x4} = -12.7

Normal stresses are obtained by dividing each normal force by the corresponding cross sectional area. Elements 1 and 2 are subjected to tensile forces and element 3 is subjected to a compressive force.

Elemental stresses	ΔT = 20°C	ΔT = 0°C
$\sigma_{(1)}$ (MPa)	23	58
$\sigma_{(2)}$ (MPa)	30	77
$\sigma_{(3)}$ (MPa)	-255	-185

Properties of the Bar Stiffness Matrix

The bar global stiffness matrix is characterized by the following:

1. Being symmetric. For instance, $K_{12} = K_{21}$.
2. Being singular. We cannot evaluate the nodal displacements of the structure unless at least one nodal displacement is known in advance as a boundary condition. From a physical point of view, this ensures that the bar would not move as a rigid body.
3. That every diagonal entry $k_{ij} \geq 0$.
4. That the summation of the coefficients of each column is equal to zero. This is useful for checking hand calculations.

An Alternative Derivation of the Element Stiffness Matrix

The following derivation is systematic and can be used easily for other types of elements. We write the unknown coefficients k_{ij} as shown below.

$$\begin{pmatrix} f_{x1} \\ f_{x2} \end{pmatrix}_e = \begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix}_e \begin{pmatrix} u_1 \\ u_2 \end{pmatrix} \qquad or \qquad (f)_e = [k]_e (u)$$

The matrix equation is valid for any combination of u_1 and u_2. Take $u_1 = 1.0$ and $u_2 = 0.0$ (Fig. 6). Then $f_{x1} = k_{11}$ and $f_{x2} = k_{21}$. However, from elementary mechanics $f_{x1} = EA / L\ u_1 = k$ and $f_{x2} = -k$. Therefore, $k_{11} = k$ and $k_{21} = -k$.

Taking $u_1 = 0$ and $u_2 = 1$ yields the expressions for the remainder coefficients ($k_{21} = -k$ and $k_{22} = k$).

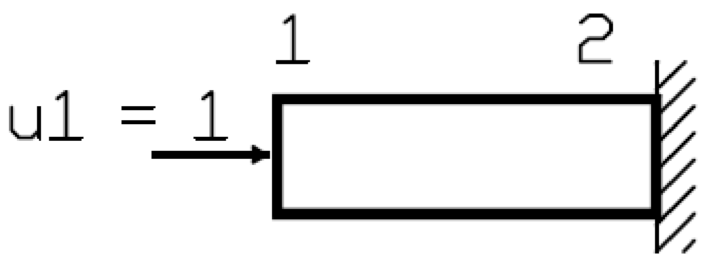


Fig. 6

Truss Elements (2-Dim)

Degrees of Freedom

The element has two nodes. Each node has two degrees of freedom, Fig. 7a. The nodal displacements and forces are shown in Figs. 7b and 7c.
The element is inclined by an angle θ. We are going to implement the following definitions; c ≡ cos θ and s ≡ sin θ.

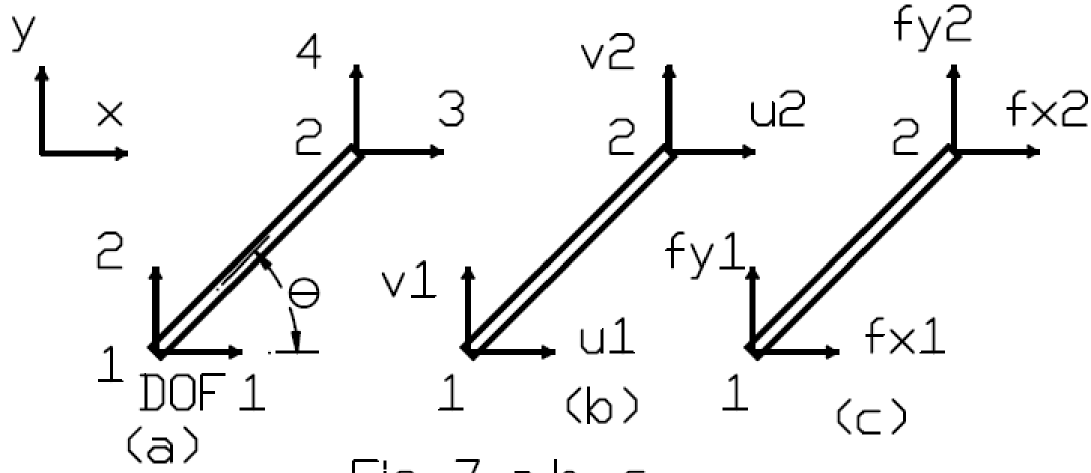


Fig. 7 a,b, c

The Element Stiffness Matrix

The matrix equation is given below.

$$\begin{pmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \end{pmatrix} = k \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \begin{pmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{pmatrix}$$

The matrix has a size of 4 x 4, because there are four degrees of freedom.
The angle θ is measured in the counter clockwise direction. Hence, θ is negative when measured in the clockwise direction. We can use θ or (θ ± 180°) and still get the same stiffness matrix.
The stiffness matrix is symmetric and singular. Diagonal terms are ≥ 0. For each column, the sum of the coefficients in odd rows (as well as those in even rows) is equal zero.

Derivation of [k]

The general expression is:

$$\begin{pmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \end{pmatrix} = k \begin{bmatrix} k_{11} & k_{12} & k_{13} & k_{14} \\ k_{21} & k_{22} & k_{23} & k_{24} \\ k_{31} & k_{32} & k_{33} & k_{34} \\ k_{41} & k_{42} & k_{43} & k_{44} \end{bmatrix} \begin{pmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{pmatrix}$$

We would first determine the coefficients of the first column. Take u_1= 1 and $v_1 = u_2 = v_2 = 0$

u_1	v_1	u_2	v_2
1	0	0	0

Then $f_{x1} = k_{11}$, $f_{y1} = k_{21}$, $f_{x2} = k_{31}$, and $f_{y2} = k_{41}$.
However, we can determine these nodal forces independently.
Due to the imposed displacement $u_1 = 1$, the bar contracts by $\delta = u_1 \cos\theta = \cos\theta$, Fig. 8. Then $f = k\,\delta = k \cos\theta \equiv k\,c$.

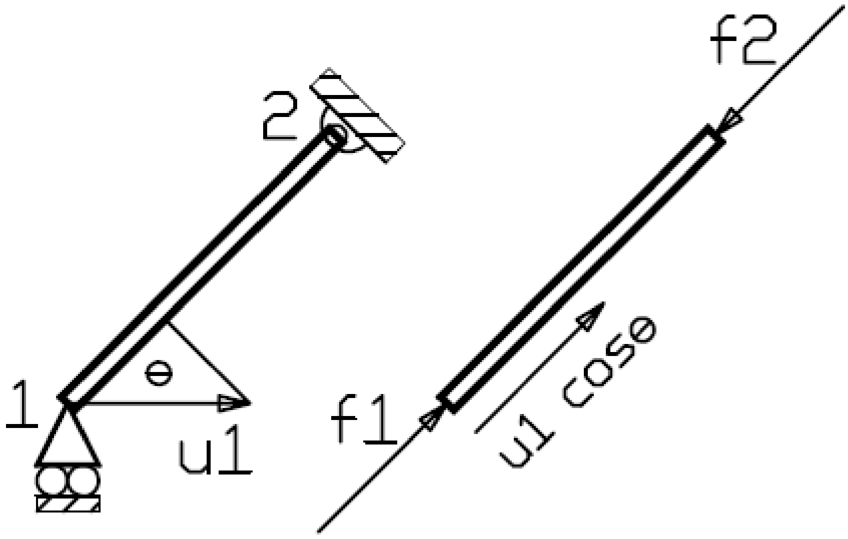


Fig. 8

The force f is inclined by an angle θ Resolve f into f_{x1} and f_{y1} to get $f_{x1} = f \cos\theta = k\,c^2$ and $f_{y1} = f \sin\theta = k\,c\,s$. Thus, $k_{11} = k\,c^2$ and $k_{12} = k\,c\,s$.
In addition, by resolving f acting at node 2, we can show that $k_{31} = - k\,c^2$ and $k_{41} = - k\,c\,s$.

We could determine the coefficients of columns 2, 3, and 4 by using the following displacement states.

Column no.	u_1	v_1	u_2	v_2
2	0	1	0	0
3	0	0	1	0
4	0	0	0	1

Example (2)

Construct the reduced stiffness matrix of the shown truss, Fig. 9. Then determine the nodal displacements and the normal stress in element 3. $L_{(1)} = L_{(2)} = 2$ m, $L_{(3)} = 2\sqrt{2}$, $A_{(1)} = A_{(2)} = A_{(3)} = 80$ mm^2, and E = 200 GPa.

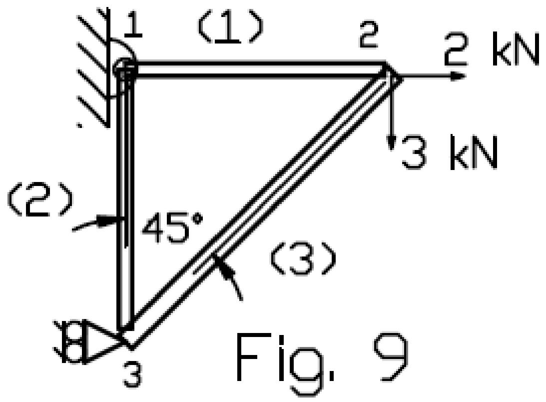


Fig. 9

Solution

Calculate the following quantities:

	Element (1)	Element (2)	Element (3)
K = EA / L N/m	8×10^6	8×10^6	5.65685×10^6
θ	0	270° (or -90°)	225° (-135 or 45)
c^2	1	0	0.5
s^2	0	1	0.5
cs	0	0	0.5

The stiffness matrix for each element:

Element (1)

$$\begin{pmatrix} f_{x1} \\ f_{y1} \\ f_{x2} \\ f_{y2} \end{pmatrix} = 10^6 \begin{bmatrix} 8 & 0 & -8 & 0 \\ 0 & 0 & 0 & 0 \\ -8 & 0 & (8) & (0) \\ 0 & 0 & (0) & (0) \end{bmatrix} \begin{pmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{pmatrix}$$

Element (2)

$$\begin{pmatrix} f_{x1} \\ f_{y1} \\ f_{x3} \\ f_{y3} \end{pmatrix} = 10^6 \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 8 & 0 & -8 \\ 0 & 0 & 0 & 0 \\ 0 & -8 & 0 & ((8)) \end{bmatrix} \begin{pmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{pmatrix}$$

Element (3)

$$\begin{pmatrix} f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{pmatrix} = 10^6 \begin{bmatrix} (((2.8284))) & (((2.8284))) & -2.8284 & (((-2.8284))) \\ (((2.8284))) & (((2.8284))) & -2.8284 & (((-2.8284))) \\ -2.8284 & -2.8284 & 2.8284 & 2.8284 \\ (((-2.8284))) & (((-2.8284))) & 2.8284 & (((2.8284))) \end{bmatrix} \begin{pmatrix} u_2 \\ v_2 \\ u_3 \\ v_3 \end{pmatrix}$$

The brackets identify the coefficients that contribute to the reduced stiffness matrix.

Boundary conditions:

$u_1 = 0$	$v_1 = 0$	$u_3 = 0$
$F_{x2} = 2000$ N	$F_{y2} = -3000$	$F_{y3} = 0$

The structure stiffness matrix has a size of 6 x 6. The reduced stiffness matrix has a size of 3 x 3. We construct the reduced stiffness matrix by ignoring the rows and columns corresponding to u_1, v_1, and u_3.

$$\begin{pmatrix} F_{x2} \\ F_{y2} \\ F_{y3} \end{pmatrix} = 10^6 \begin{bmatrix} (8)+2.8284 & (0)+2.8284 & -2.8284 \\ (0)+2.8284 & (0)+2.8284 & -2.8284 \\ -2.8284 & -2.8284 & ((8))+2.8284 \end{bmatrix} \begin{pmatrix} u_2 \\ v_2 \\ v_3 \end{pmatrix} =$$

$$10^6 \begin{bmatrix} 10.8284 & 2.8284 & -2.8284 \\ 2.8284 & 2.8284 & -2.8284 \\ -2.8284 & -2.8284 & 10.8284 \end{bmatrix} \begin{pmatrix} u_2 \\ v_2 \\ v_3 \end{pmatrix}$$

Solve to get the required nodal displacements.

u_2 (m)	v_2	v_3
0.625×10^{-3}	-2.061×10^{-3}	-0.375×10^{-3}

Calculate the nodal forces acting on element (3);

$$\begin{pmatrix} f_{x2} \\ f_{y2} \\ f_{x3} \\ f_{y3} \end{pmatrix} = 2.8284.10^6 \begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix} \begin{pmatrix} u_2 = 0.625\,(10^{-3}) \\ v_2 = -2.061\,(10^{-3}) \\ u_3 = 0 \\ v_3 = -0.375\,(10^{-3}) \end{pmatrix} = \begin{pmatrix} -3\,10^3 \\ -3\,10^3 \\ 3\,10^3 \\ 3\,10^3 \end{pmatrix} \quad N$$

The resultant of the elemental nodal forces acting on node 2

$f_2 = \sqrt{\left((-3\,10^3)^2 + (-3\,10^3)^2 \right)} = 4.24$ kN; and
$f_3 = \sqrt{\left((3\,10^3)^2 + (3\,10^3)^2 \right)} = 4.24$ kN

These forces are compressive as shown in Fig. 10.
The normal stress is
$\sigma_{x\,(3)} = -\,(4.24 \times 10^{3}) / (80 \times 10^{-6}) = -53 \times 10^{3}$ Pa = -53 MPa *Ans.*

Fig. 10 shows the forces acting on the other elements. (Try to calculate Them.)

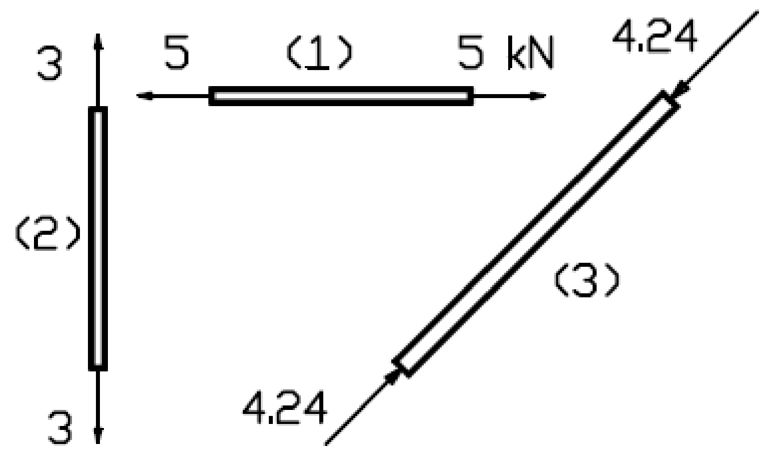


Fig. 10

Beam Elements (2-Dim)

We are going to deal with a 2-dim horizontal beam subjected to transverse and bending moments only.

Degrees of Freedom

Fig. 11 shows a beam element. It has two degrees of freedom per node. The element stiffness matrix has a size of 4 x 4. The sign convention used for the moments and forces is not universal.

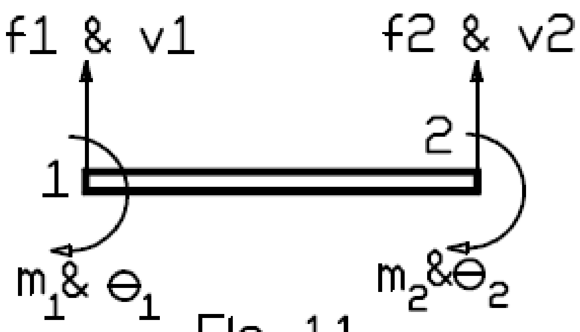


Fig. 11

The Stiffness Matrix

The matrix is:

$$\begin{pmatrix} f_1 \\ m_1 \\ f_2 \\ m_2 \end{pmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & -6L & -12 & -6L \\ -6L & 4L^2 & 6L & 2L^2 \\ -12 & 6L & 12 & 6L \\ -6L & 2L^2 & 6L & 4L^2 \end{bmatrix}\begin{pmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{pmatrix}$$

Where, I is the centroidal second moment of area about the z axis ($I \equiv I_z$). This matrix equation is valid only when $I_{yz} = 0$. We should use another type of elements when the y z axes are not principal axes.

An Outline of How to Derive [k]

The stiffness matrix could be derived by calculating the response of the beam to specific independent states of displacements similar to the approach used for deriving the truss element stiffness.

Example (3)

Get the vertical deflection and angle of rotation (slope) at node 2, (Fig. 12). Get the results in terms of E, I, and L.

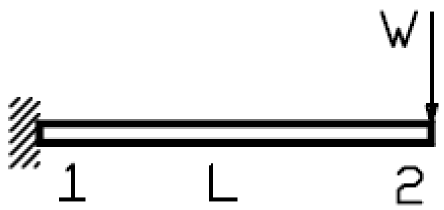


Fig. 12

Solution

The boundary conditions are:

$v_1 = 0$	$\theta_1 = 0$
$F_2 = -W$	$M_2 = 0$

By ignoring the 1st and 2nd columns and rows in the element stiffness matrix, we get the following matrix equation.

$$\begin{pmatrix} -W \\ 0 \end{pmatrix} = \frac{EI}{L^3}\begin{bmatrix} 12 & 6L \\ 6L & 4L^2 \end{bmatrix}\begin{pmatrix} v_2 \\ \theta_2 \end{pmatrix}$$

We solve these simultaneous equations to get

$v_2 =$	$(W L^3 / 3EI)$
$\Theta_2 =$	$(W L^2 / 2EI)$

The student should verify these results. Note: the slope at node 2 is clockwise.

Distributed Loads

Fig. 13-a shows a uniformly distributed force w (N/m). This force is replaced by equivalent nodal loads as shown in Fig. 13-b (consult a textbook for the proof).

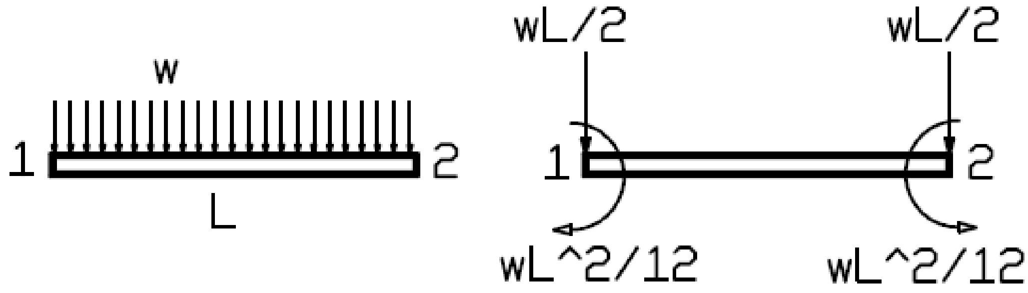


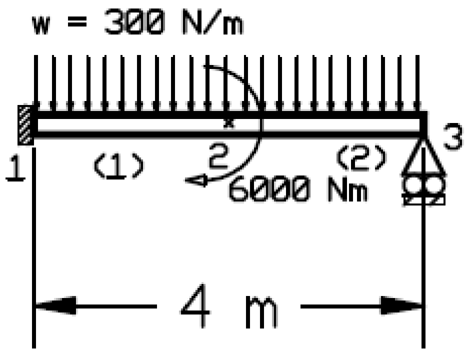
Fig. 13-(a) Fig. 13-(b)

The element equation is:

$$\begin{pmatrix} f_1 \\ m_1 \\ f_2 \\ m_2 \end{pmatrix} + \begin{pmatrix} \frac{-wl}{2} \\ \frac{wl^2}{12} \\ \frac{-wl}{2} \\ \frac{-wl^2}{12} \end{pmatrix} = [K](\delta)$$

Where, $\{\delta\}^T = \{ v_1 \quad \theta_1 \quad v_2 \quad \theta_2 \}^T$.

Example (4)

Construct the reduced system of equations for the shown beam. Then determine the nodal displacements and rotations. Moreover, calculate the nodal loads acting on each element. $I_z = 4 \times 10^{-6}$ m^4 and E = 200 GPa.	 Fig. 14

Solution

We shall model the beam using two elements. Each has L = 2 m.

$EI / L^3 = (200 \times 10^9)(4 \times 10^{-6}) / 2^3 = 10^5$ N/m	$wL/2 = (300)(2) / 2 = 300$ N	$wL^2/12 = 100$ Nm

The elements equations

Element (1)

$$\begin{pmatrix} f_1 \\ m_1 \\ f_2 \\ m_2 \end{pmatrix}_{(1)} + \begin{pmatrix} -300 \\ 100 \\ -300 \\ -100 \end{pmatrix} = 10^5 \begin{bmatrix} 12 & -12 & -12 & -12 \\ -12 & 16 & 12 & 8 \\ -12 & 12 & 12 & 12 \\ -12 & 8 & 12 & 16 \end{bmatrix} \begin{pmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{pmatrix}$$

Element (2)

$$\begin{pmatrix} f_2 \\ m_2 \\ f_3 \\ m_3 \end{pmatrix}_{(2)} + \begin{pmatrix} -300 \\ 100 \\ -300 \\ -100 \end{pmatrix} = 10^5 \begin{bmatrix} 12 & -12 & -12 & -12 \\ -12 & 16 & 12 & 8 \\ -12 & 12 & 12 & 12 \\ -12 & 8 & 12 & 16 \end{bmatrix} \begin{pmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{pmatrix}$$

Boundary Conditions

$v_1 = \theta_1 = v_3 = 0$	$F_2 = 0$	$M_2 = 6000$ Nm	$M_3 = 0$

The reduced system of equations

$$\begin{pmatrix} 0 \\ 6000 \\ 0 \end{pmatrix} + \begin{pmatrix} -300-300 \\ -100+100 \\ -100 \end{pmatrix} = 10^5 \begin{bmatrix} 12+12 & 12-12 & -12 \\ 12-12 & 16+16 & 8 \\ -12 & 8 & 16 \end{bmatrix} \begin{pmatrix} v_2 \\ \theta_2 \\ \theta_3 \end{pmatrix}$$

or

$$\begin{pmatrix} -600 \\ 6000 \\ -100 \end{pmatrix} = 10^5 \begin{bmatrix} 24 & 0 & -12 \\ 0 & 32 & 8 \\ -12 & 8 & 16 \end{bmatrix} \begin{pmatrix} v_2 \\ \theta_2 \\ \theta_3 \end{pmatrix}$$

Solve the system of equations to get:

v_2 = -0.0014375 m = -1.438 mm	Θ_2 = 0.00246875 rad = 0.1414°	Θ_3 = -.002375 rad = -0.1361°

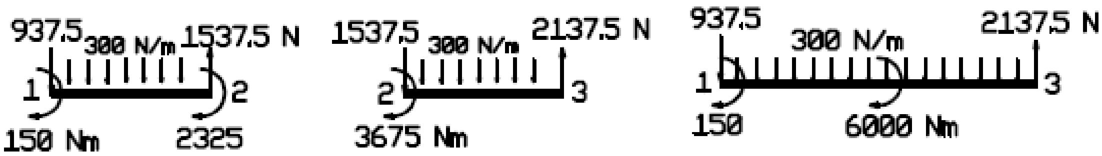
The loads acting on element (1)

$$\begin{pmatrix} f_1 \\ m_1 \\ f_2 \\ m_2 \end{pmatrix}_{(1)} = 10^5 \begin{bmatrix} 12 & -12 & -12 & -12 \\ -12 & 16 & 12 & 8 \\ -12 & 12 & 12 & 12 \\ -12 & 8 & 12 & 16 \end{bmatrix} \begin{pmatrix} 0 \\ 0 \\ -0.0014375 \\ 0.00246875 \end{pmatrix} + \begin{pmatrix} 300 \\ -100 \\ 300 \\ 100 \end{pmatrix} = \begin{pmatrix} -937.5 \\ 150 \\ 1537.5 \\ 2325 \end{pmatrix}$$

Element (2)

$$\begin{pmatrix} f_2 \\ m_2 \\ f_3 \\ m_3 \end{pmatrix}_{(2)} = 10^5 \begin{bmatrix} 12 & -12 & -12 & -12 \\ -12 & 16 & 12 & 8 \\ -12 & 12 & 12 & 12 \\ -12 & 8 & 12 & 16 \end{bmatrix} \begin{pmatrix} v_2 \\ \theta_2 \\ 0 \\ \theta_3 \end{pmatrix} + \begin{pmatrix} 300 \\ -100 \\ 300 \\ 100 \end{pmatrix} = \begin{pmatrix} -1537.5 \\ 3675 \\ 2137.5 \\ 0 \end{pmatrix}$$

The nodal loads acting on the elements are shown in Figs. 15-(a) & 15-(b). The reaction loads acting on the beam are shown in Fig. 15-(c).



Element (1) Fig. 15-(a) Element (2) Fig. 15-(b) The beam Fig. 15-(c)

Document 2 contains the computer results to this very same problem. The computer solution gives not only the nodal displacements but also the entire elastic curve.

Symmetry

The following table depicts examples of symmetric beams and trusses under static conditions Symmetry is in geometry, material properties, relevant boundary conditions, as well as in loading. Each configuration has a plane of symmetry. This plane virtually cut the structure into two identical parts. Therefore, we could reduce the size of the problem by half. In doing this, we should introduce the proper boundary conditions at the plane of symmetry (the new edge of the reduced structure).

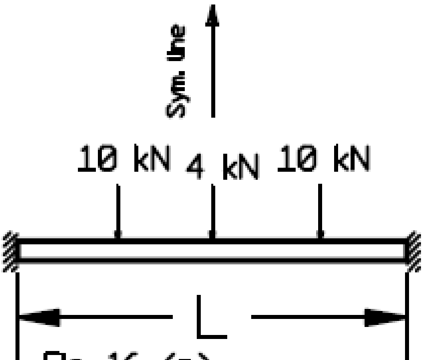
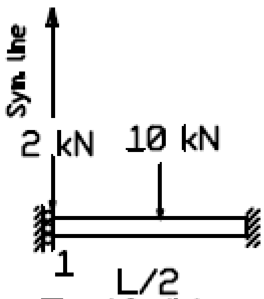
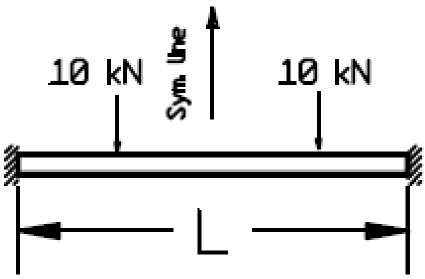
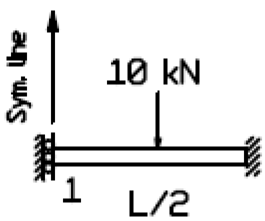
For beams:

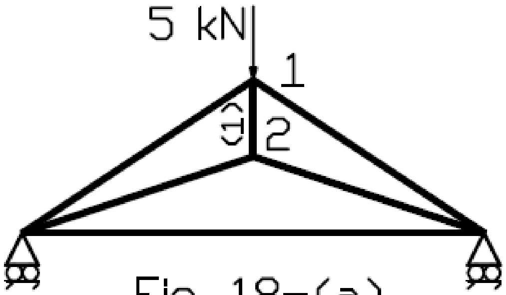
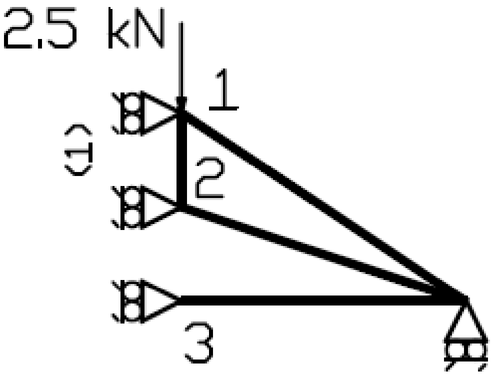
- The slope at the plane of symmetry θ is zero.
- The transverse force acting along the plane of symmetry must be halved.

For trusses:

- The displacement at the plane of symmetry normal to it u is zero.
- The forces at this cutting plane must be halved.
- The cross sectional area of bars aligned with the axis of symmetry must be halved.
- Bars that cross the plane of symmetry at an angle must be cut by that plane resulting in a shorter bar (and u = 0 at the intersection).

Figures 16a, 17a, and 18a can be modelled by Figs. 16b to 18b.

Configuration	Min # of elements	B.C. at axis of symmetry
 Fig. 16-(a)	4	F = -4 kN M = 0 {θ_1 = 0. However, this should not be used as a boundary condition.}
 Fig. 16-(b)	2	F_1 = -2 kN θ_1 = 0
 Fig. 17-(a)	3	F = 0 ; M = 0 We may solve the problem without a node in the middle.
 Fig. 17-(b)	2	F_1 = 0 θ_1 = 0

Configuration	Min # of elements	B.C. at axis of symmetry
 Fig. 18-(a)	6	$F_{x1} = 0$ $F_{y1} = -5$ kN $F_{x2} = 0$ $F_{y2} = 0$ Note: $A_{(1)}$ = **1000** mm^2
 Fig. 18-(b)	4	$u_1 = 0$ $F_{y1} = -2.5$ kN $u_2 = 0$ $F_{y2} = 0$ Plus $u_3 = 0$ $F_3 = 0$ Note: $A_{(1)}$ = **500** mm^2
Configuration	Min # of elements	B.C. at axis of symmetry

Plane Frames

Fig. 19 shows a simple planar frame with assigned nodes and elements (or joints and members). Each element is capable of sustaining bending moments, shearing and axial forces.
A typical plane frame element (Fig. 20) has two nodes each has three degrees of freedom.
The element equation is:

$\{f\}_{(e)} = [k]_{(e)} \{\delta\}$; where

$\{f\}^T = \{ f_{x1} \ \ f_{y1} \ \ m_1 \ \ f_{x2} \ \ f_{y2} \ \ m_2 \}^T$ and
$\{\delta\}^T = \{ u_1 \ \ v_1 \ \ \theta_1 \ \ u_2 \ \ v_2 \ \ \theta_2 \}^T$

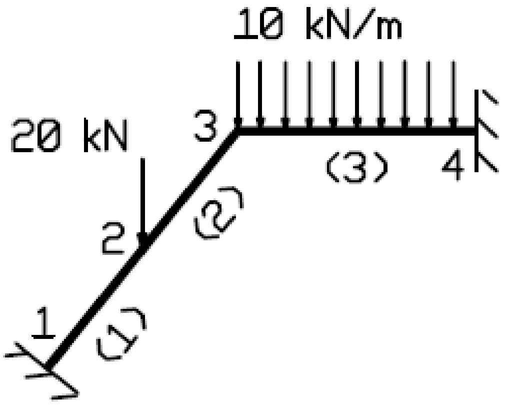


Fig. 19

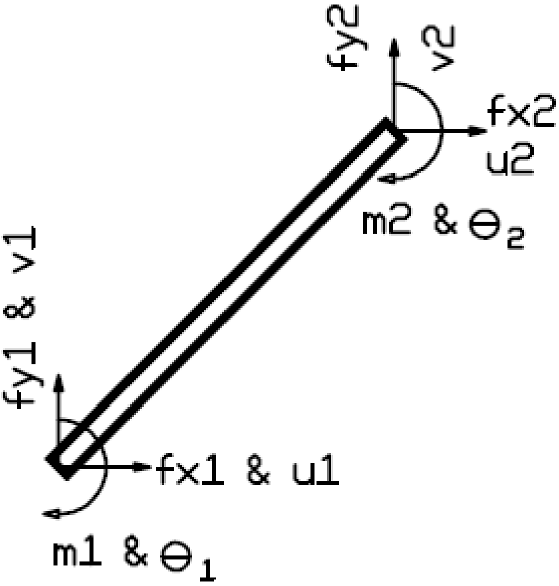


Fig. 20

Global Versus Local Axes

Fig. 21 shows two sets of axes x-y of the whole structure (or global axes) and x`-y` local axes. The axis x` is aligned with the centroidal axis of the member. The local axes are useful in inputting distributed forces perpendicular to inclined elements.

Frames are sometimes made of segments connected by hinges as for the linkages of a shoe brake (Fig. 22). Therefore frame elements may have one node hinged but the other node transmits moment (Fig. 23). Moreover, computer programs allow the user to input both nodes of a frame element as hinges. Hence, a frame element can be used as a truss element.

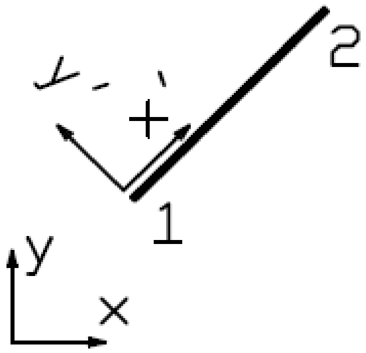


Fig. 21

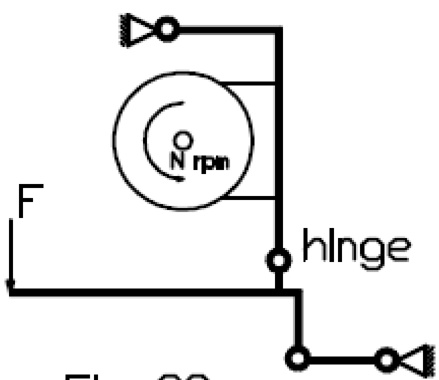


Fig. 22

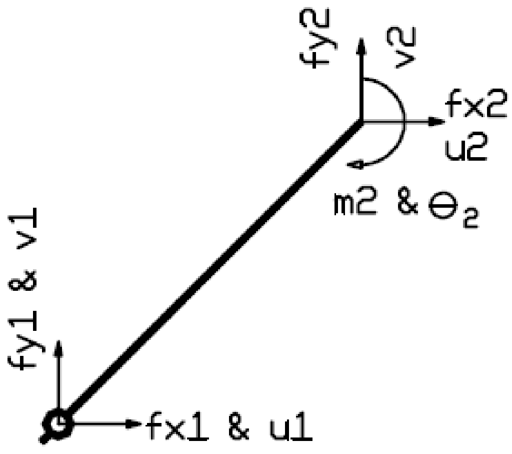


Fig. 23

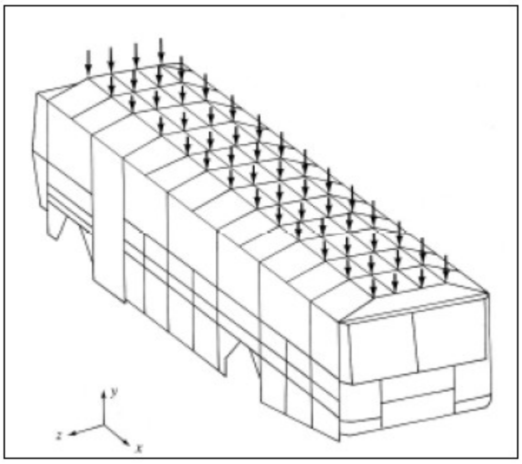


Fig. 24

A Practical Example

Fig. 24 shows a bus frame subjected to roof load of 100 kN in order to test its strength. This is an example of how engineers use computer programs to solve engineering problems (Logan). In this example, 599 frame elements and 357 nodes were used.

Comments

This introduction is elementary and limited in scope. Many topics were omitted such as inclined rolling supports and the details of frame elements.

In order to appreciate the strength of the method, students should solve certain assigned problems using a computer program.

References

P.P. Benham, R.J. Crawford & C.G. Armstrong (1996) *Mechanics of Engineering Materials*, 2nd edition, Longman, Essex.
R.G. Budynas (1999) *Advanced Strength and Applied Stress Analysis*, 2nd edition, McGraw-Hill, Boston.
H. Grandin,Jr. (1986) *Fundamentals of the Finite Element Method*, Macmillan, New York.
C.E. Knight (1993) *The Finite Element Method in Mechanical Design*, PWS-Kent, Boston.
R. L. Logan (1992) *A first course in the Finite Element method*, PWS-Kent, Boston.
S. Moaveni (2003) *Finite Element Analysis – Theory and Application with ANSYS*, 2nd edition, Pearson Education / Prentice Hall, New Jersey. (www.prenhall.com/Moaveni)

Appendix – Formulas

1-Dim bar

$$EA\alpha\,\Delta T\begin{pmatrix}-1\\1\end{pmatrix}+\begin{pmatrix}f_{x1}\\f_{x2}\end{pmatrix}_e=\begin{bmatrix}k & -k\\-k & k\end{bmatrix}_e\begin{pmatrix}u_1\\u_2\end{pmatrix}$$

Fig. 2-a

Plane truss

$$\begin{pmatrix}f_{x1}\\f_{y1}\\f_{x2}\\f_{y2}\end{pmatrix}=k\begin{bmatrix}c^2 & cs & -c^2 & -cs\\cs & s^2 & -cs & -s^2\\-c^2 & -cs & c^2 & cs\\-cs & -s^2 & cs & s^2\end{bmatrix}\begin{pmatrix}u_1\\v_1\\u_2\\v_2\end{pmatrix}$$

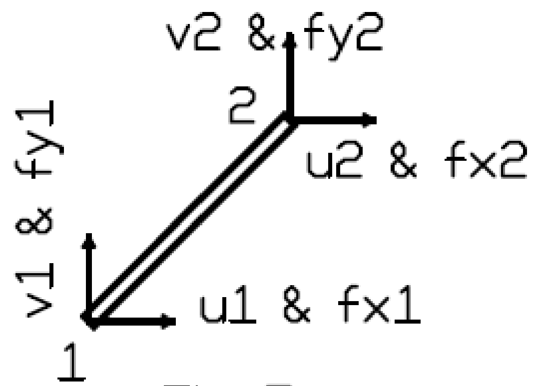


Fig. 7

Plane horizontal beam

$$\begin{pmatrix}f_1\\m_1\\f_2\\m_2\end{pmatrix}=\frac{EI}{L^3}\begin{bmatrix}12 & -6L & -12 & -6L\\-6L & 4L^2 & 6L & 2L^2\\-12 & 6L & 12 & 6L\\-6L & 2L^2 & 6L & 4L^2\end{bmatrix}\begin{pmatrix}v_1\\\theta_1\\v_2\\\theta_2\end{pmatrix}$$

$$\begin{pmatrix}f_1\\m_1\\f_2\\m_2\end{pmatrix}+\begin{pmatrix}\frac{-wl}{2}\\\frac{wl^2}{12}\\\frac{-wl}{2}\\\frac{-wl^2}{12}\end{pmatrix}=[K](\delta)$$

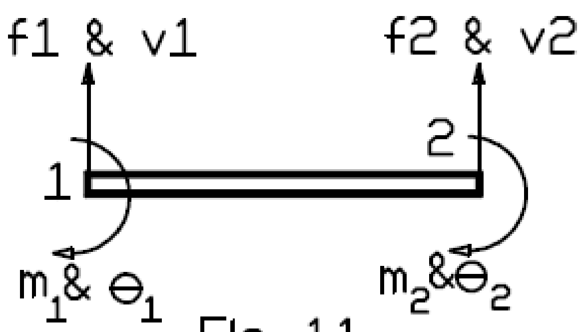


Fig. 11