LECTURE NOTES

ON

ADVANCED STRUCTURAL ANALYSIS

(BSTB01)

M.Tech - I Semester (IARE-R16)

PREPARED BY

Dr. VENU M

PROFESSOR

Department of Civil Engineering

INSTITUTE OF AERONAUTICAL ENGINEERING

(Autonomous)

Dundigal – 500 043, Hyderabad.

UNIT-I
INFLUENCE COEFFICIENTS

INTRODUCTION:

Indeterminate structures are being widely used for its obvious merits. It may be recalled that, in the case of indeterminate structures either the reactions or the internal forces cannot be determined from equations of statics alone. In such structures, the number of reactions or the number of internal forces exceeds the number of static equilibrium equations. In addition to equilibrium equations, compatibility equations are used to evaluate the unknown reactions and internal forces in statically indeterminate structure. In the analysis of indeterminate structure it is necessary to satisfy the equilibrium equations (implying that the structure is in equilibrium) compatibility equations (requirement if for assuring the continuity of the structure without any breaks) and force displacement equations (the way in which displacement are related to forces). We have two distinct method of analysis for statically indeterminate structure depending upon how the above equations are satisfied:

1. Force method of analysis (also known as flexibility method of analysis, method of consistent deformation, flexibility matrix method)

2. Displacement method of analysis (also known as stiffness matrix method).

In the force method of analysis, primary unknown are forces. In this method compatibility equations are written for displacement and rotations (which are calculated by force displacement equations). Solving these equations, redundant forces are calculated. Once the redundant forces are calculated, the remaining reactions are evaluated by equations of equilibrium. In the displacement method of analysis, the primary unknowns are the displacements. In this method, first force -displacement relations are computed and subsequently equations are written satisfying the equilibrium conditions of the structure. After determining the unknown displacements, the other forces are calculated satisfying the compatibility conditions and force displacement relations. The displacement-based method is amenable to computer programming and hence the method is being widely used in the modern day structural analysis. In general, the maximum deflection and the maximum stresses are small as compared to statically determinate structure.

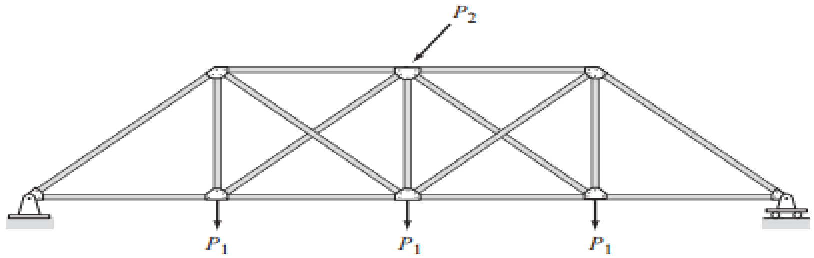
Two different methods can be used for the matrix analysis of structures: the flexibility method, and the stiffness method. The flexibility method, which is also referred to as the force or compatibility method, is essentially a generalization in matrix form of the classical method of consistent deformations. In this approach, the primary unknowns are the redundant forces, which are calculated first by solving the structure's compatibility equations. Once the redundant forces are known, the displacements can be evaluated by applying the equations of equilibrium and the appropriate member force–displacement relations.

CLASSIFICATION OF FRAMED STRUCTURES

Framed structures are composed of straight members whose lengths are significantly larger than their cross-sectional dimensions. Common framed structures can be classified into six basic categories based on the arrangement of their members, and the types of primary stresses that may develop in their members under major design loads.

Plane Trusses

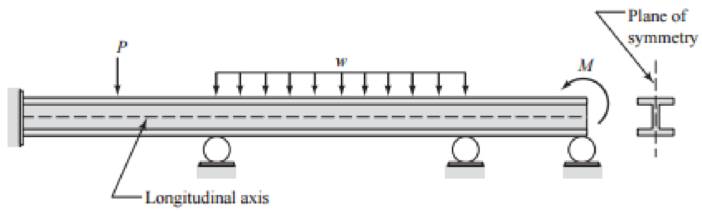
A truss is defined as an assemblage of straight members connected at their ends by flexible connections, and subjected to loads and reactions only at the joints (connections). The members of such an ideal truss develop only axial forces when the truss is loaded. In real trusses, such as those commonly used for supporting roofs and bridges, the members are connected by bolted or welded connections that are not perfectly flexible, and the dead weights of the members are distributed along their lengths. Because of these and other deviations from idealized conditions, truss members are subjected to some bending and shear. However, in most trusses, these secondary bending moments and shears are small in comparison to the primary axial forces, and are usually not considered in their designs. If large bending moments and shears are anticipated, then the truss should be treated as a rigid frame (discussed subsequently) for analysis and design. If all the members of a truss as well as the applied loads lie in a single plane, the truss is classified as a plane truss. The members of plane trusses are assumed to be connected by frictionless hinges. The analysis of plane trusses is considerably simpler than the analysis of space (or three-dimensional) trusses. Fortunately, many commonly used trusses, such as bridge and roof trusses, can be treated as plane trusses for analysis.



Plane Truss

Beams

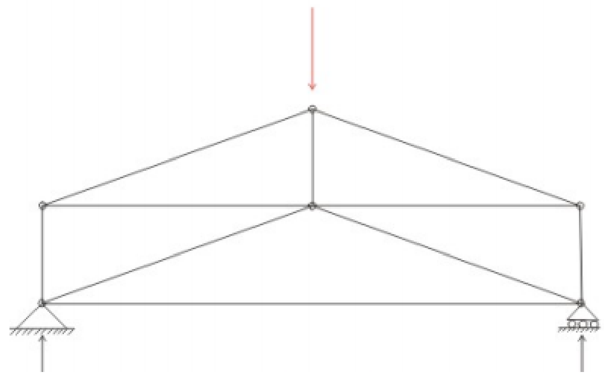
A beam is defined as a long straight structure that is loaded perpendicular to its longitudinal axis. Loads are usually applied in a plane of symmetry of the beam's cross-section, causing its members to be subjected only to bending moments and shear forces.



Beam

Space Trusses

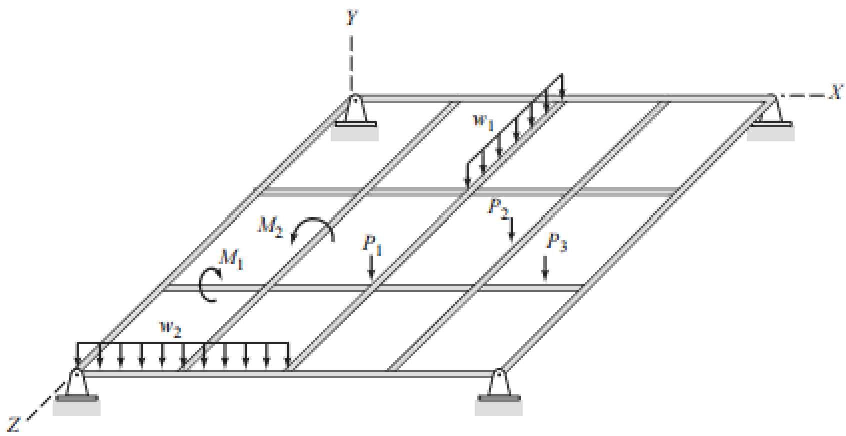
Some trusses (such as lattice domes, transmission towers, and certain aerospace structures cannot be treated as plane trusses because of the arrangement of their members or applied loading. Such trusses, referred to as space trusses, are analyzed as three-dimensional structures subjected to three dimensional force systems. The members of space trusses are assumed to be connected by frictionless ball-and-socket joints, and the trusses are subjected to loads and reactions only at the joints. Like plane trusses, the members of space trusses develop only axial forces.



Space Trusses

Grids

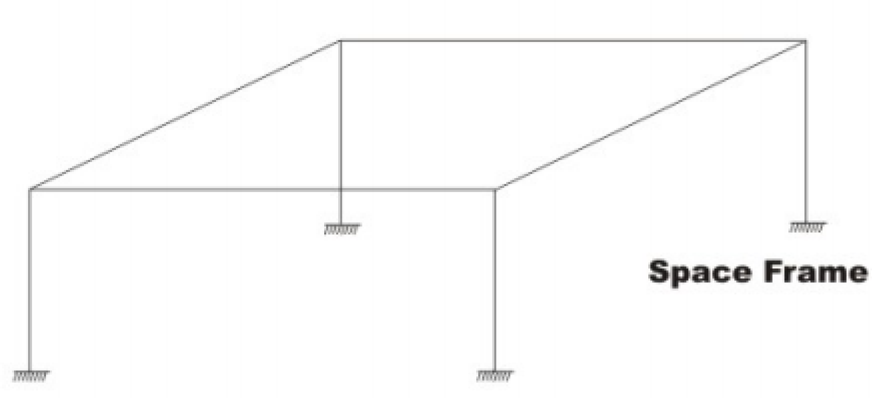
A grid, like a plane frame, is composed of straight members connected together by rigid and/or flexible connections to form a plane framework. The main difference between the two types of structures is that plane frames are loaded in the plane of the structure, whereas the loads on grids are applied in the direction perpendicular to the structure's plane. Members of grids may, therefore, be subjected to torsional moments, in addition to the bending moments and corresponding shears that cause the members to bend out of the plane of the structure. Grids are commonly used for supporting roofs covering large column-free areas in such structures as sports arenas, auditoriums, and aircraft hangars.



Grid

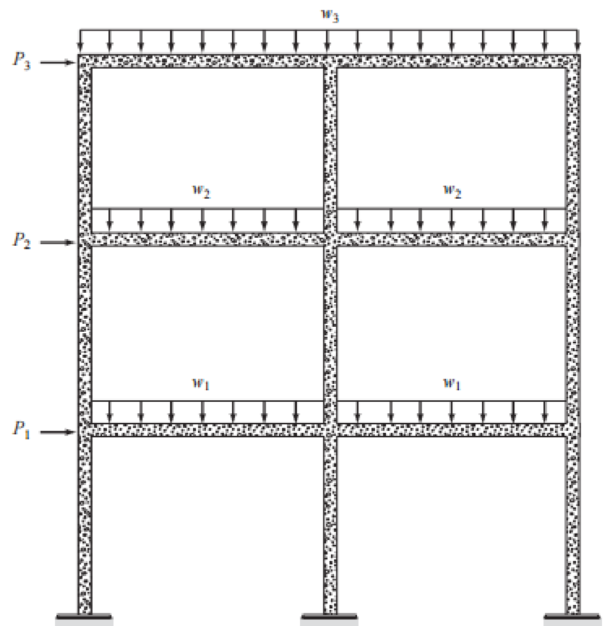
Space Frames

Space frames constitute the most general category of framed structures. Members of space frames may be arranged in any arbitrary directions, and connected by rigid and/or flexible connections. Loads in any directions may be applied on members as well as on joints. The members of a space frame may, in general, be subjected to bending moments about both principal axes, shears in principal directions, torsional moments, and axial forces.



Plane Frames

Frames, also referred to as rigid frames, are composed of straight members connected by rigid (moment resisting) and/or flexible connections. Unlike trusses, which are subjected to external loads only at the joints, loads on frames may be applied on the joints as well as on the members. If all the members of a frame and the applied loads lie in a single plane, the frame is called a plane frame. The members of a plane frame are, in general, subjected to bending moments, shears, and axial forces under the action of external loads. Many actual three-dimensional building frames can be subdivided into plane frames for analysis.



Plane Frame

FUNDAMENTAL RELATIONSHIPS FOR STRUCTURAL ANALYSIS

Structural analysis, in general, involves the use of three types of relationships:

- Equilibrium equations,
- compatibility conditions, and
- constitutive relations.

Equilibrium Equation

A structure is considered to be in equilibrium if, initially at rest, it remains at rest when subjected to a system of forces and couples. If a structure is in equilibrium, then all of its members and joints must also be in equilibrium. Recall from statics that for a plane (two-dimensional) structure lying in the XY plane and subjected to a coplanar system of forces and couples, the necessary and sufficient conditions for equilibrium can be expressed in Cartesian (XY) coordinates. These equations are referred to as the equations of equilibrium for plane structures. For a space (three-dimensional) structure subjected to a general three dimensional system of forces and couples (Fig. 1.12),

the equations of equilibrium are expressed as

FX = 0 , FY = 0 and FZ = 0

MX = 0, MY = 0 and MZ = 0

For a structure subjected to static loading, the equilibrium equations must be satisfied for the entire structure as well as for each of its members and joints. In structural analysis, equations of equilibrium are used to relate the forces (including couples) acting on the structure or one of its members or joints.

Compatibility Conditions

The compatibility conditions relate the deformations of a structure so that its various parts (members, joints, and supports) fit together without any gaps or overlaps. These conditions (also referred to as the continuity conditions) ensure that the deformed shape of the structure is continuous (except at the locations of any internal hinges or rollers), and is consistent with the support conditions. Consider, for example, the two-member plane frame. The deformed shape of the frame due to an arbitrary loading is also depicted, using an exaggerated scale. When analysing a structure, the compatibility conditions are used to relate member end displacements to joint displacements which, in turn, are related to the support conditions. For example, because joint 1 of the frame is attached to a roller support that cannot translate in the vertical direction, the vertical displacement of this joint must be zero. Similarly, because joint 3 is attached to a fixed support that can neither rotate nor translate in any direction, the rotation and the horizontal and vertical displacements of joint 3 must be zero.

GLOBAL AND LOCAL COORDINATE SYSTEMS

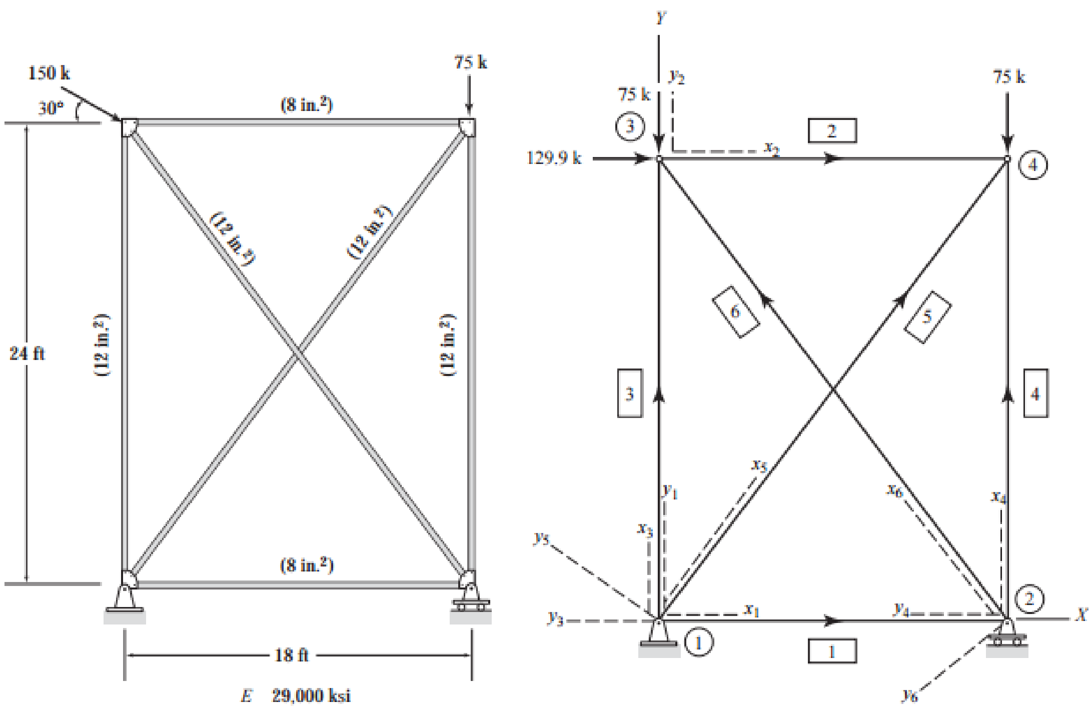
In the matrix stiffness method, two types of coordinate systems are employed to specify the structural and loading data and to establish the necessary force–displacement relations. These are referred to as the global (or structural) and the local (or member) coordinate systems.

Global Coordinate System

The overall geometry and the load–deformation relationships for an entire structure are described with reference to a Cartesian or rectangular global coordinate system. When analyzing a plane (two-dimensional) structure, the origin of the global XY coordinate system can be located at any point in the plane of the structure, with the X and Y axes oriented in any mutually perpendicular directions in the structure's plane. However, it is usually convenient to locate the origin at a lower left joint of the structure, with the X and Y axes oriented in the horizontal (positive to the right) and vertical (positive upward) directions, respectively, so that the X and Y coordinates of most of the joints are positive.

Local Coordinate System

Since it is convenient to derive the basic member force–displacement relationships in terms of the forces and displacements in the directions along and perpendicular to members, a local coordinate system is defined for each member of the structure.



(a) Actual Truss

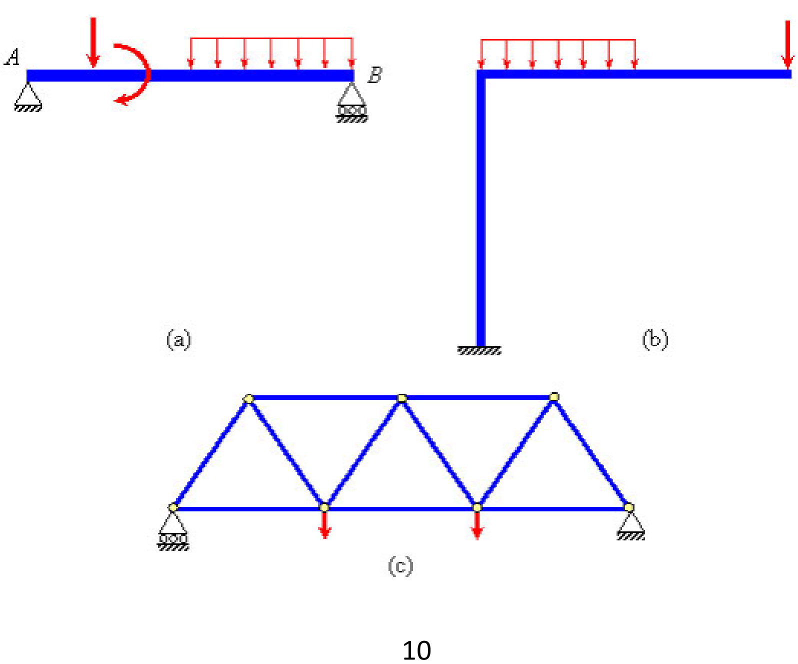
(b) Analytical Model Showing Global and Local Coordinate Systems

DEGREES OF FREEDOM

The degrees of freedom of a structure, in general, are defined as the independent joint displacements (translations and rotations) that are necessary to specify the deformed shape of the structure when subjected to an arbitrary loading. Since the joints of trusses are assumed to be frictionless hinges, they are not subjected to moments and, therefore, their rotations are zero. Thus, only joint translations must be considered in establishing the degrees of freedom of trusses. The deformed shape of the truss, for an arbitrary loading, is depicted in using an exaggerated scale. From this figure, we can see that joint 1, which is attached to the hinged support, cannot translate in any direction; therefore, it has no degrees of freedom. Because joint 2 is attached to the roller support, it can translate in the X direction, but not in the Y direction. Thus, joint 2 has only one degree of freedom, which is designated d1 in the figure. As joint 3 is not attached to a support, two displacements (namely, the translations d2 and d3 in the X and Y directions, respectively) are needed to completely specify its deformed position 3 . Thus, joint 3 has two degrees of freedom. Similarly, joint 4, which is also a free joint, has two degrees of freedom, designated d4 and d5.

Static Indeterminacy of Structures

If the number of independent static equilibrium equations (refer to Section 1.2) is not sufficient for solving for all the external and internal forces (support reactions and member forces, respectively) in a system, then the system is said to be statically indeterminate. A statically determinate system, as against an indeterminate one, is that for which one can obtain all the support reactions and internal member forces using only the static equilibrium equations. For example, idealized as one-dimensional, the number of independent static equilibrium equations is just 1 while the total number of unknown support reactions are 2, that is more than the number of equilibrium equations available. Therefore, the system is considered statically indeterminate. The following figures illustrate some example of statically determinate and indeterminate structures.

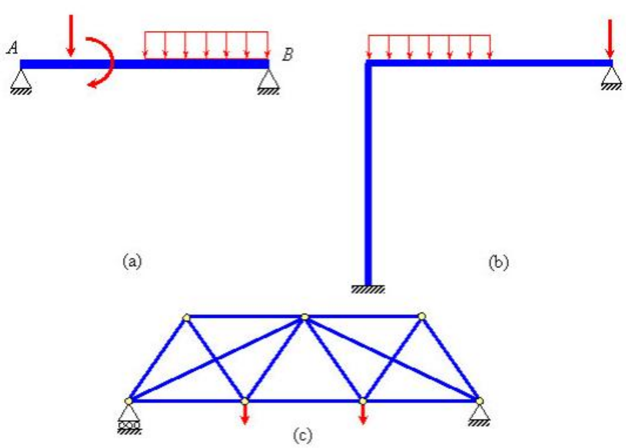


the equilibrium equations are described as the necessary and sufficient conditions to maintain the equilibrium of a body. However, these equations are not always able to provide all the information needed to obtain the unknown support reactions and internal forces. The number of external supports and internal members in a system may be more than the number that is required to maintain its equilibrium configuration. Such systems are known as indeterminate systems and one has to use compatibility conditions and constitutive relations in addition to equations of equilibrium to solve for the unknown forces in that system. For an indeterminate system, some support(s) or internal member(s) can be removed without disturbing its equilibrium. These additional supports and members are known as redundants . A determinate system has the exact number of supports and internal members that it needs to maintain the equilibrium and no redundants. If a system has less than required number of supports and internal members to maintain equilibrium, then it is considered unstable . For example, the two-dimensional propped cantilever system in (Figure 1.13a) is an indeterminate system because it possesses one support more than that are necessary to maintain its equilibrium. If we remove the roller support at end B (Figure 1.13b), it still maintains equilibrium. One should note that here it has the same number of unknown support reactions as the number of independent static equilibrium equations.

$$\sum F_x = 0$$

$$\sum F_y = 0$$

$$\sum M_z (\text{about any point}) = 0$$



Statically indeterminate structures

An indeterminate system is often described with the number of redundants it posses and this number is known as its degree of static indeterminacy . Thus, mathematically:

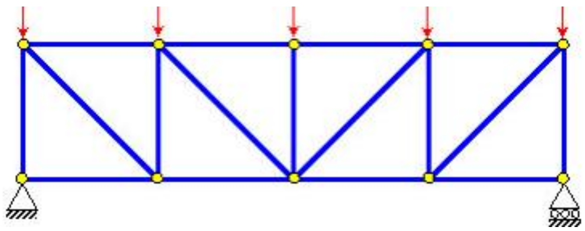
Degree of static indeterminacy = Total number of unknown (external and internal) forces - Number of independent equations of equilibrium

It is very important to know exactly the number of unknown forces and the number of independent equilibrium equations. Let us investigate the determinacy/indeterminacy of a few two-dimensional pin-jointed truss systems. Let m be the number of members in the truss system and n be the number of pin (hinge) joints connecting these members. Therefore, there will be m number of unknown internal forces (each is a two-force member) and 2 n numbers of independent joint equilibrium equations (and for each joint, based on its free body diagram). If the support reactions involve r unknowns, then:

Total number of unknown forces = m + r

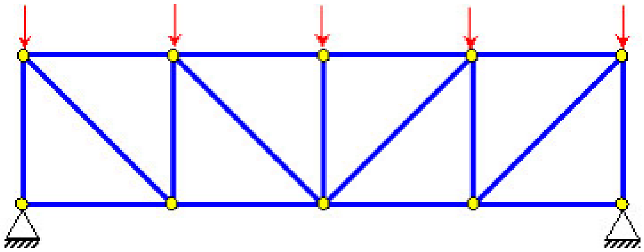
Total number of independent equilibrium equations = 2 n

So, **degree of static indeterminacy = (m + r) - 2 n**



Determinate truss

m = 17, n = 10, and r = 3. So, degree of static indeterminacy = 0, that means it is a statically determinate system.



(Internally) indeterminate truss

m = 18, n = 10, and r = 3. So, degree of static indeterminacy = 1.

Kinematic Indeterminacy of Structures

A structure is said to be kinematically indeterminate if the displacement components of its joints cannot be determined by compatibility conditions alone. In order to evaluate displacement components at the joints of these structures, it is necessary to consider the equations of static equilibrium. i.e. no. of unknown joint displacements over and above the compatibility conditions will give the degree of kinematic indeterminacy.

We have seen that the degree of statical indeterminacy of a structure is, in fact, the number of forces or stress resultants which cannot be determined using the equations of statical equilibrium. Another form of the indeterminacy of a structure is expressed in terms of its *degrees of freedom*; this is known as the *kinematic indeterminacy, n*k, of a structure and is of particular relevance in the stiffness method of analysis where the unknowns are the displacements.

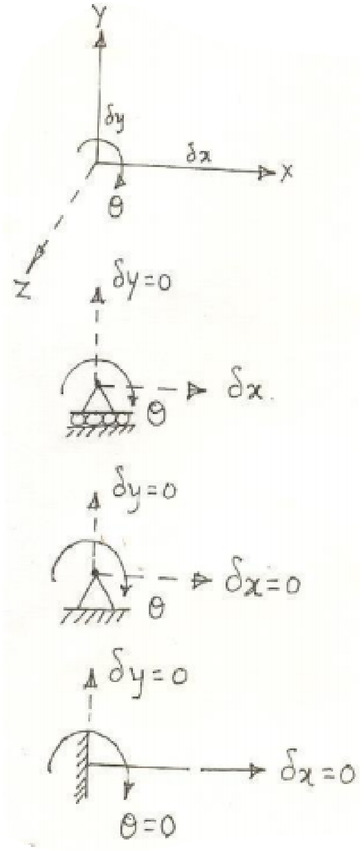
A simple approach to calculating the kinematic indeterminacy of a structure is to sum the degrees of freedom of the nodes and then subtract those degrees of freedom that are prevented by constraints such as support points. It is therefore important to remember that in three-dimensional structures each node possesses 6 degrees of freedom while in plane structures each node possess three degrees of freedom.

For determinate structures, the force method allows us to find internal forces (using equilibrium i.e. based on Statics) irrespective of the material information. Material (stress-strain) relationships are needed only to calculate deflections. However, for indeterminate structures , Statics (equilibrium) alone is not sufficient to conduct structural analysis. Compatibility and material information are essential.

Fixed beam :

Kinematically determinate :

Simply supported beam Kinematically indeterminate



Any joint – Moves in three directions in a plane structure Two displacements δx in x direction, δy in y direction, θ rotation about z axis as shown.

Roller Support :
r = 1, $\delta y = 0$, θ & δx exist – DOF = 2 e = 1

Hinged Support :
r = 2, $\delta x = 0$, $\delta y = 0$, θ exists – DOF = 1 e = 2

Fixed Support :
r = 3, $\delta x = 0$, $\delta y = 0$, $\theta = 0$ DOF = 0 e = 3

Reaction components prevent the displacements no. of restraints = no. of reaction components.

Degree of kinematic indeterminacy:

Pin jointed structure: Every joint – two displacements components and no rotation

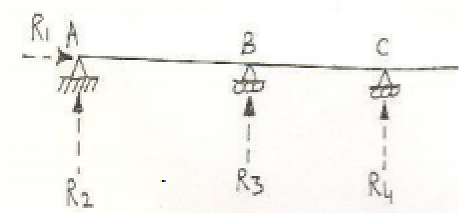
$\therefore$ **Dk = 2j – e** where, e = no. of equations of compatibility
= no. of reaction components

Rigid Jointed Structure:

Every joint will have three displacement components, two displacements and one rotation. Since, axial force is neglected in case of rigid jointed structures, it is assumed that the members are inextensible & the conditions due to inextensibility of members will add to the numbers of restraints. i.e to the ‘e’ value.

∴ **Dk = 3j – e** where, e = no. of equations of compatibility
= no. of reaction components +
constraints due to in extensibility

Example 1 : Find the static and kinematic indeterminacies
r = 4, m = 2, j = 3



3j

= (3 x 2 + 4) – 3 x 3 = 1

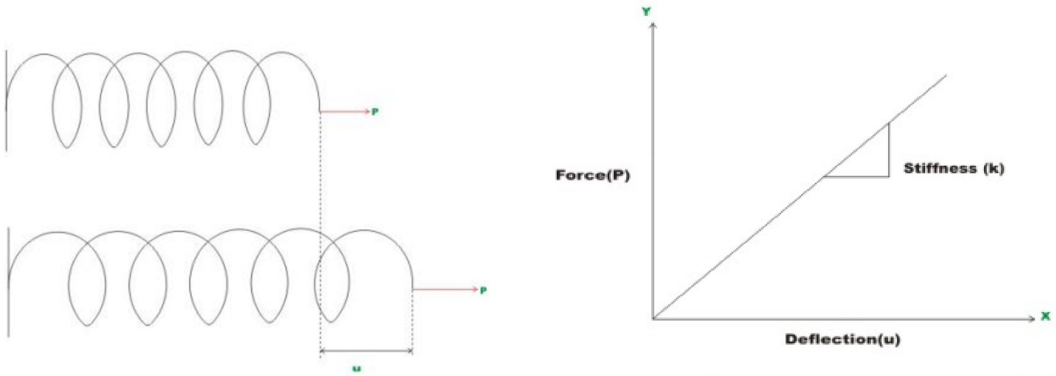
Dk = 3j – e

= 3 x 3 – 6 = 3

i.e. rotations at A,B, & C i.e. θa, θb & θc are the displacements.

(e = reaction components + inextensibility conditions = 4 + 2 = 6)

Force-Displacement Relationship



Consider linear elastic spring as shown in Fig. Let us do a simple experiment. Apply a force at the end of spring and measure the deformation . Now increase the load to and measure the deformation . Likewise repeat the experiment for different values of load . Result may be represented in the form of a graph as shown in the above figure where load is shown on -axis and deformation on abscissa. The slope of this graph is known as the stiffness of the spring and is represented by and is given by

$$k = \frac{P_2 - P_1}{u_2 - u_1} = \frac{P}{u}$$

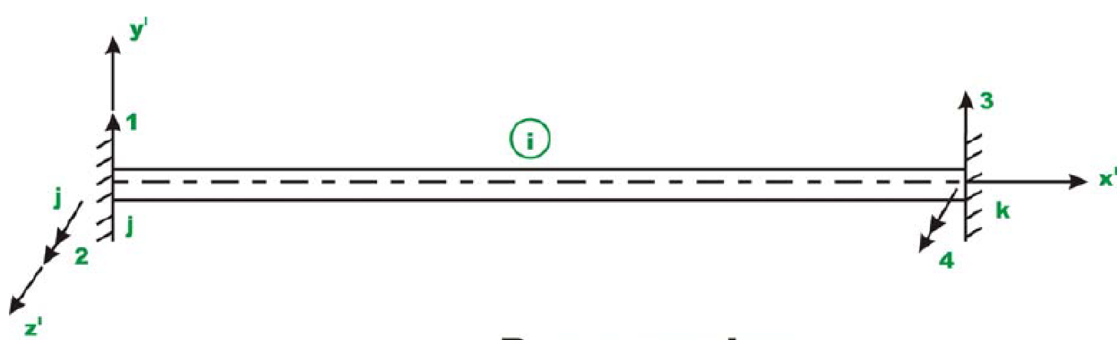
$$P = ku$$

The spring stiffness may be defined as the force required for the unit deformation of the spring. The stiffness has a unit of force per unit elongation. The inverse of the stiffness is known as flexibility. It is usually denoted by and it has a unit of displacement per unit force.

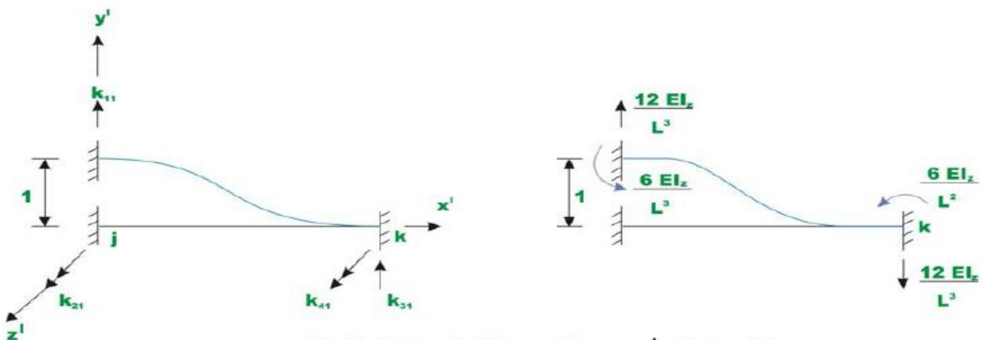
$$a = \frac{1}{k} \quad P = ku$$

UNIT – II
STIFFNESS METHOD APPLIED TO LARGE FRAMES

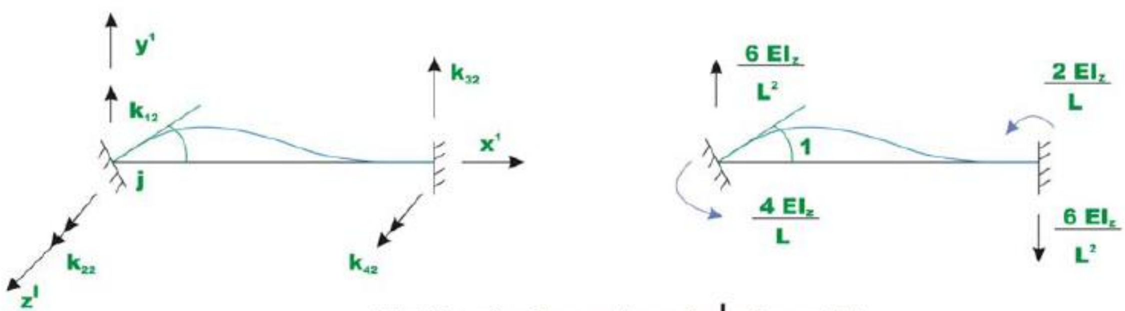
Two degrees of freedom (one translation and one rotation) are considered at each end of the member. Hence, there are four possible degrees of freedom for this member and hence the resulting stiffness matrix is of the order 4x4. In this method counterclockwise moments and counterclockwise rotations are taken as positive. The positive sense of the translation and rotation are also shown in the figure. Displacements are considered as positive in the direction of the coordinate axis. The elements of the stiffness matrix indicate the forces exerted on the member by the restraints at the ends of the member when unit displacements are imposed at each end of the member. Let us calculate the forces developed in the above beam member when unit displacement is imposed along each degree of freedom holding all other displacements to zero. Now impose a unit displacement along y' axis at j end of the member while holding all other displacements to zero. This displacement causes both shear and moment in the beam. The restraint actions are also shown in the figure. By definition they are elements of the member stiffness matrix. In particular they form the first column of element stiffness matrix. In Fig., the unit rotation in the positive sense is imposed at j end of the beam while holding all other displacements to zero.



Beam member



Unit translation along y' at end j



Unit rotation about z' at end j

unit displacement along y' axis at end k is imposed and corresponding restraint actions are calculated. Similarly in Fig. , unit rotation about z' axis at end k is imposed and corresponding stiffness coefficients are calculated. Hence the member stiffness matrix for the beam member is

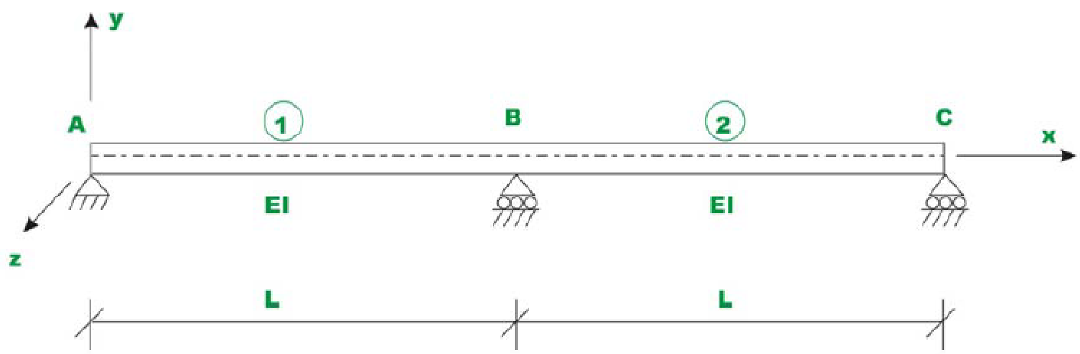
$$
[k] = \begin{array}{c} \\ \\ \end{array}
\begin{array}{cccc} 1 & 2 & 3 & 4 \end{array}
$$

$$
[k] = \left[\begin{array}{cc|cc}
\frac{12EI_z}{L^3} & \frac{6EI_z}{L^2} & -\frac{12EI_z}{L^3} & \frac{6EI_z}{L^2} \\
\frac{6EI_z}{L^2} & \frac{4EI_z}{L} & -\frac{6EI_z}{L^2} & \frac{2EI_z}{L} \\
\hline
-\frac{12EI_z}{L^3} & -\frac{6EI_z}{L^2} & \frac{12EI_z}{L^3} & -\frac{6EI_z}{L^2} \\
\frac{6EI_z}{L^2} & \frac{2EI_z}{L} & -\frac{6EI_z}{L^2} & \frac{4EI_z}{L}
\end{array}\right]
\begin{array}{c} \underline{1} \\ \underline{2} \\ \underline{3} \\ \underline{4} \end{array}
$$

The stiffness matrix is symmetrical. The stiffness matrix is partitioned to separate the actions associated with two ends of the member. For continuous beam problem, if the supports are unyielding, then only rotational degree of freedom is possible. In such a case the first and the third rows and columns will be deleted. The reduced stiffness matrix will be, Beam (global) Stiffness Matrix.

$$[k]=\left[\begin{array}{c|c}\dfrac{4EI_z}{L} & \dfrac{2EI_z}{L} \\ \hline \dfrac{2EI_z}{L} & \dfrac{4EI_z}{L}\end{array}\right]$$

The formation of structure (beam) stiffness matrix from its member stiffnessmatrices is explained with help of two span continuous beam shown in Fig. Note that no loading is shown on the beam. The orthogonal co-ordinatesystem *xyz* denotes the global co-ordinate system.



Continuous beam

Assembly of Stiffness Matrix and Force Vector

After the evaluation of element stiffness matrix and element force vector for all the elements, these quantities need to be " **assembled** " to get the global stiffness matrix and global force vector. As stated at the end of section 6.2, this procedure has two steps:

Expansion of the element stiffness matrix and element force vector to the full size.

i. Addition of the expanded matrices and vectors over all the elements. At this stage, the second term of the expression for $\{F\}$ (equation 6.8) also needs to be added.

Let us first discuss the first step. Note that equations (6.25) and (6.26) are the expressions for the element stiffness matrix $[k^{(k)}]$ and the element force vector $\{f^{(k)}\}$ while equations (6.21) and (6.22) are the expressions for their expanded versions $[K^{(k)}]$ and $\{F^{(k)}\}$. When we compare equations (6.25) with (6.21), we observe that (1,1) component of $[k^{(k)}]$ occupies the position (k,k) of the expanded matrix $[K^{(k)}]$. This is because k is the global number of the local node 1 of the element k. Thus, the first step involves:

- Choose the component $k_{ij}^{(k)}$, $i = 1$, $j = 1$
- Find the global number of the local nodes i and j of the element k. Let they be r and s respectively.
- Then the component $k_{ij}^{(k)}$ occupies the location in r -th row and s -th column of the expanded matrix $[K^{(k)}]$. Thus, the component $k_{ij}^{(k)}$ goes to the location $K_{rs}^{(k)}$ in the expanded matrix.

Repeat the steps (i)-(iii) for the other values of i and j. The remaining components of $[K^{(k)}]$ are made zero.

The first step can be expressed mathematically by introducing a matrix $[C]$, called as the **connectivity matrix** , which relates the local and global numbering systems. The number of rows in the connectivity matrix is equal to the number of elements and the number of columns is equal to the number of nodes per element. Thus, the row index of $[C]$ denotes the element number and the column index of $[C]$ represents the local node number. The elements of $[C]$ are the corresponding global node numbers. Thus, for the mesh of Fig. 6.1, the connectivity matrix becomes

$$[C] = \begin{bmatrix} 1 & 2 \\ 2 & 3 \\ 3 & 4 \\ - & - \\ - & - \\ k & k+1 \\ - & - \\ - & - \\ - & - \\ N & N+1 \end{bmatrix} kth-row \tag{6.34}$$

The first row of the connectivity matrix contains the global numbers of the first and second local nodes of element 1. The global numbers corresponding to the first and second local nodes of element 2 are written in the second row. Continuing in this way, the global numbers of the first and second local nodes of element k appear in the k-th row. The last row contains the global numbers associated with the first and second local nodes of the last, i.e. N-th element. The expression (6.34), in the index notation, can be expressed as

$$r = C_{ki} \tag{6.35}$$

It means the global number r of the local node i of the element k is obtained as the value of the component of the connectivity matrix in k-th row and i-th column. As an example, consider the case of k = 3 and i = 2. The expression (6.35) gives $r = C_{32} = 4$. This means 4 is the global number of the second local node of the element 3. This can be verified from Fig. 6.1.

Now, the first step of the assembly procedure can be expresses as follows. The expanded matrix $[K^{(k)}]$ is obtained from the element stiffness matrix $[k^{(k)}]$ by the relation:

$$\begin{aligned} K_{rs}^{(k)} &= k_{ij}^{(k)} \qquad where \quad r = C_{ki},\ s = C_{kj}; \\ &= 0 \qquad otherwise. \end{aligned} \tag{6.36}$$

Similarly, to obtain the expanded vector $\{F^{(k)}\}$ from the element force vector $\{f^{(k)}\}$, we use the relation:

$$F_r^k = f_i^k \quad where\ r = C_{ki};$$
$$= 0 \quad otherwise. \tag{6.37}$$

Thus, we use the following procedure:

i. Choose the component $f_i^k, i = 1$
ii. Find the global number of the local node i from the connectivity matrix. Let it be r.
iii. Then, the component f_i^k goes to the location F_r^k of the expanded matrix.
iv. Repeat the steps (i)-(iii) for the other values of i. The remaining components of $\{F^k\}$ are made zero.

The second-step is straight-forward. After obtaining the expanded versions of the element stiffness matrix and the element force vector for all the elements, they are added as follows:

$$[K] = \sum_{k=1}^{N} [K^{(k)}], \tag{6.38}$$

$$\{F\} = \sum_{k=1}^{N} \{F^{(k)}\} + \{P\} \tag{6.39}$$

The matrix $\{P\}$ corresponds to the second term of equation (6.8). Note that, the only basis function which is nonzero at $x = L$ is ϕ_{N+1}. Further, it's value at $x = L$ is 1. Thus

$$P\phi_i \Big|_{x=L} = 0 \qquad for\ i = 1,2,\ldots N$$
$$= P \qquad for\ i = N+1 \tag{6.40}$$

Therefore, the vector {P} can be written as

$$\{P\} = \begin{Bmatrix} 0 \\ - \\ - \\ - \\ - \\ 0 \\ P \end{Bmatrix} \begin{matrix} 1st-row \\ \\ \\ \\ \\ Nth-row \\ (N+1)th-row \end{matrix} \tag{6.41}$$

Example on Assembly of Stiffness Matrix and Force Vector

As an example, consider the mesh of 6 elements (N = 6) and 7 nodes, shown in Fig. 6.4.

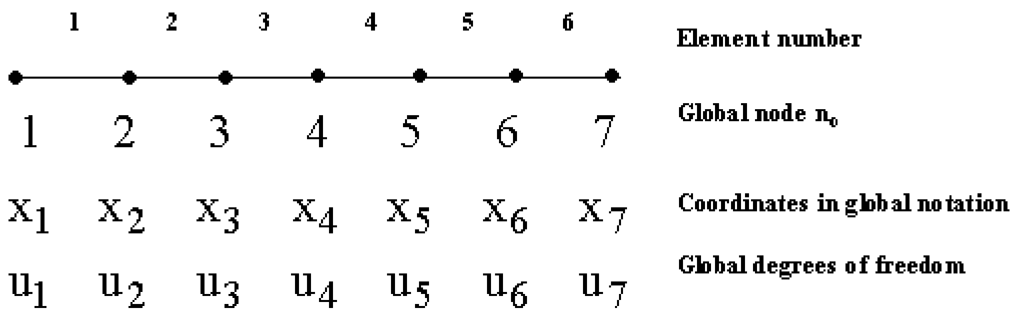


Figure 6.4 Mesh with 6 elements

The connectivity matrix for this mesh can be written as:

$$[C] = \begin{bmatrix} 1 & 2 \\ 2 & 3 \\ 3 & 4 \\ 4 & 5 \\ 5 & 6 \\ 6 & 7 \end{bmatrix}. \tag{6.42}$$

Let

$$[k^{(k)}] = \begin{bmatrix} k_{11}^{(k)} & k_{12}^{(k)} \\ k_{21}^{(k)} & k_{22}^{(k)} \end{bmatrix}, \tag{6.43}$$

And

$$\{f^{(k)}\} = \begin{Bmatrix} f_1^{(k)} \\ f_2^{(k)} \end{Bmatrix} \tag{6.44}$$

be the element stiffness matrix and the element force vector of the elements k =1,2,3,4,5,6.

Consider the element 1, i.e. $k = 1$. Note that

$$\begin{aligned} &(i)\ \ i = 1, j = 1 \ gives \ \ r = C_{ki} = C_{11} = 1, s = C_{kj} = C_{11} = 1; \\ &(ii)\ i = 1, j = 2 \ \ gives \ \ r = C_{ki} = C_{11} = 1, s = C_{kj} = C_{12} = 2; \\ &(iii)\ i = 2, j = 1 \ gives \ \ r = C_{ki} = C_{12} = 2, s = C_{kj} = C_{11} = 1; \\ &(iv)\ i = 2, j = 2 \ \ gives \ \ r = C_{ki} = C_{12} = 2, s = C_{kj} = C_{12} = 2; \end{aligned} \tag{6.45}$$

Then as per equation (6.36), components of the stiffness matrix of the element 1, i.e. of $[k^{(1)}]$, occupy the following locations in the expanded matrix $[\mathrm{k}^{(1)}]$:

$$k_{11}^{(1)} \to K_{11}^{(1)}, \quad k_{12}^{(1)} \to K_{12}^{(1)}, \ k_{21}^{(1)} \to K_{21}^{(1)}, \ k_{22}^{(1)} \to K_{22}^{(1)} \tag{6.46}$$

Similarly, as per equation (6.37), components of the force vector of the element 1, i.e. of $\{f^{(k)}\}$, occupy the following locations in the expanded vector $\{F^{(k)}\}$:

$$f_1^{(1)} \to F_1^{(1)}, \quad f_2^{(1)} \to F_2^{(1)} \tag{6.47}$$

The remaining components of the expanded matrix $[K^{(1)}]$ and the expanded vector $\{F^{(1)}\}$ are zero. Thus, the matrix $[K^{(1)}]$ becomes:

$$[K^{(1)}] = \begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & 0 & 0 & 0 & 0 & 0 \\ k_{21}^{(1)} & k_{22}^{(1)} & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \qquad (6.48)$$

.

and the vector $\{F^{(1)}\}$ becomes:

$$\{F^{(1)}\} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} \qquad (6.49)$$

Similarly, we obtain the expanded versions of the element stiffness matrix $[k^{(k)}]$ and the element force vector $\{f^{(k)}\}$ for the remaining elements, i.e. for k = 2,3,4,5,6. It can easily be verified that, for the 3 rd element (i.e. for $k = 3$), the expanded matrix $[K^{(3)}]$ and the expanded vector $[F^{(3)}]$ are:

$$[K^{(3)}] = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & k_{11}^{(3)} & k_{12}^{(3)} & 0 & 0 & 0 \\ 0 & 0 & k_{21}^{(3)} & k_{22}^{(3)} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \qquad (6.50a)$$

$$\{F^{(3)}\} = \begin{Bmatrix} 0 \\ 0 \\ f_1^{(3)} \\ f_2^{(3)} \\ 0 \\ 0 \\ 0 \end{Bmatrix} \tag{6.50b}$$

This completes the first step.

In the 2nd step, we add all the expanded matrices and vectors. Thus, equation (6.38) gives the following expression for the global stiffness matrix:

$$[K] = \sum_{k=1}^{6} [K^{(k)}] = \begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & 0 & 0 & 0 & 0 & 0 \\ k_{21}^{(1)} & k_{22}^{(1)} + k_{11}^{(2)} & k_{12}^{(2)} & 0 & 0 & 0 & 0 \\ 0 & k_{22}^{(2)} & k_{22}^{(2)} + k_{11}^{(3)} & k_{12}^{(3)} & 0 & 0 & 0 \\ 0 & 0 & k_{21}^{(3)} & k_{22}^{(3)} + k_{11}^{(4)} & k_{12}^{(4)} & 0 & 0 \\ 0 & 0 & 0 & k_{21}^{(4)} & k_{22}^{(2)} + k_{11}^{(5)} & k_{12}^{(5)} & 0 \\ 0 & 0 & 0 & 0 & k_{21}^{(5)} & k_{22}^{(5)} + k_{11}^{(6)} & k_{12}^{(6)} \\ 0 & 0 & 0 & 0 & 0 & k_{21}^{(6)} & k_{22}^{(6)} \end{bmatrix} \tag{6.51}$$

Similarly, the sum of the expanded force vector becomes:

$$\sum_{k=1}^{6} \{F^{(k)}\} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} + f_1^{(2)} \\ f_2^{(2)} + f_1^{(3)} \\ f_2^{(3)} + f_1^{(4)} \\ f_2^{(4)} + f_1^{(5)} \\ f_2^{(5)} + f_1^{(6)} \\ f_2^{(6)} \end{Bmatrix} \tag{6.52}$$

However, before we get the global force vector $\{F\}$, we need to add the vector $\{P\}$ to the above expression. Since N (no. of elements) = 6, the $(N+1)$-th component, i.e. the 7-th component of the vector $\{P\}$ will be P. The remaining components will be zero as per equation (6.41). Thus, $\{P\}$ becomes:

$$\{P\} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ 0 \\ P \end{Bmatrix} \tag{6.53}$$

Substituting the expressions (6.52) and (6.53) in equation (6.39), we get the following expression for the global force vector $\{F\}$:

$$\{F\} = \sum_{k=1}^{6} \{F^{(k)}\} + \{P\} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} + f_1^{(2)} \\ f_2^{(2)} + f_1^{(3)} \\ f_2^{(3)} + f_1^{(4)} \\ f_2^{(4)} + f_1^{(5)} \\ f_2^{(5)} + f_1^{(6)} \\ f_2^{(6)} + \mathrm{P} \end{Bmatrix} \tag{6.54}$$

Now, as in section 6.3, assume that EA and f (distributed force) are constant for the entire bar. Further, assume that the length h_k of each element is constant. Let us denote it by h. Then

$$h_k \equiv h = \frac{L}{N} \tag{6.55}$$

Then, equation (6.32) implies that the element stiffness matrix $[k^{(k)}]$ is identical for each element and is given by

$$[k^{(k)}] = \frac{EA}{h}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad for \quad k = 1,2,3,4,5,6 \tag{6.56}$$

Similarly, equation (6.33) implies that the element force vector $\{f^{(k)}\}$ is identical for each element and is given by

$$\{f^{(k)}\} = \frac{f_0 h}{2}\begin{Bmatrix}1\\1\end{Bmatrix} \quad for \quad k = 1,2,3,4,5,6 \tag{6.57}$$

Substituting the expression (6.56) in equation (6.51), we get

$$[K] = \frac{EA}{h}\begin{bmatrix} 1 & -1 & 0 & 0 & 0 & 0 & 0 \\ -1 & 1+1 & -1 & 0 & 0 & 0 & 0 \\ 0 & -1 & 1+1 & -1 & 0 & 0 & 0 \\ 0 & 0 & -1 & 1+1 & -1 & 0 & 0 \\ 0 & 0 & 0 & -1 & 1+1 & -1 & 0 \\ 0 & 0 & 0 & 0 & -1 & 1+1 & -1 \\ 0 & 0 & 0 & 0 & 0 & -1 & 1 \end{bmatrix}$$

$$= \frac{EA}{h}\begin{bmatrix} 1 & -1 & 0 & 0 & 0 & 0 & 0 \\ -1 & 2 & -1 & 0 & 0 & 0 & 0 \\ 0 & -1 & 2 & -1 & 0 & 0 & 0 \\ 0 & 0 & -1 & 2 & -1 & 0 & 0 \\ 0 & 0 & 0 & -1 & 2 & -1 & 0 \\ 0 & 0 & 0 & 0 & -1 & 2 & -1 \\ 0 & 0 & 0 & 0 & 0 & -1 & 1 \end{bmatrix} \tag{6.58}$$

Further, substituting the expression (6.57) in equation (6.54), we get

$$\{F\} = \frac{f_0 h}{2}\begin{Bmatrix} 1 \\ 2 \\ 2 \\ 2 \\ 2 \\ 2 \\ 1 + \dfrac{2P}{f_0 h} \end{Bmatrix} \tag{6.59}$$

In actual calculations, the assembly procedure is appropriately modified to reduce the computational time and storage requirements. When, the number of elements is large, storing of the expanded matrices and vectors for each element needs a lot of storage requirement. Therefore, the process is modified as follows:

- Once the expanded version $[K^{(1)}]$ of the element stiffness matrix of the first element $(k = 1)$ is obtained, the element stiffness matrices of other elements are not expanded.

- Instead, the locations of the components of the stiffness matrix $[k^{(2)}]$ of the element two $(k = 2)$ are determined using equation (6.36).

- From the connectivity (6.34), it is easy to see that

$$
\begin{aligned}
k_{11}^{(2)} &\rightarrow (2,2)\ \textit{location of the}\ \exp\ \textit{anded matrix}, \\
k_{12}^{(2)} &\rightarrow (2,3)\ \textit{location of the}\ \exp\ \textit{anded matrix}, \\
k_{21}^{(2)} &\rightarrow (3,2)\ \textit{location of the}\ \exp\ \textit{anded matrix}, \\
k_{22}^{(2)} &\rightarrow (3,3)\ \textit{location of the}\ \exp\ \textit{anded matrix}.
\end{aligned}
\qquad (6.60)
$$

UNIT – III
STIFFNESS MATRIX ASSEMBLY OF STRUCTURES FORCE METHOD AND APPLICATIONS TO SIMPLE PROBLEMS

Structure as a whole or any substructure Must Satisfy

1.Equilibrium of forces. 2. Displacement compatibility. 3. Force-displacement relation.

Matrix Force Method – also called as Flexibility method. Member forces are treated as the basic unknowns. Similar to the classical force method, but based on matrix approach.

S.No.	Type of displacement, Δ	Flexibility, δ	Stiffness, k
1.	Axial	$\frac{L}{AE}$	$\frac{AE}{L}$
2.	Transverse		
	(a) Far-end fixed	$\frac{L^3}{12EI}$	$\frac{12EI}{L^3}$
	(b) Far-end hinged	$\frac{L^3}{3EI}$	$\frac{3EI}{L^3}$
3.	Bending or flexural		
	(a) Far-end fixed	$\frac{L}{4EI}$	$\frac{4EI}{L}$
	(b) Far-end hinged	$\frac{L}{3EI}$	$\frac{3EI}{L}$
4.	Torsional	$\frac{L}{GK}$	$\frac{GK}{L}$

Step	*Force method (flexibility or compatibility method)*	*Displacement method (stiffness or equilibrium method)*
1.	Determine the degree of static indeterminacy (degree of redundancy), n.	Determine the degree of kinematic indeterminacy, (degree of freedom), n.
2.	Choose the redundants.	Identify the independent displacement components.
3.	Assign coordinates 1, 2, ..., n to the redundants.	Assign coordinates 1, 2, ..., n to the independent displacement components.
4.	Remove all the redundants to obtain the released structure.	Prevent all the independent displacement components to obtain the restrained structure.
5.	Determine $[\Delta_L]$, the displacements at the coordinates due to the applied loads acting on the released structure.	Determine $[P']$, the forces required at the coordinates in the restrained structure due to the loads other than those acting at the coordinates.
6.	Determine $[\Delta_R]$, the displacements at the coordinates due to the redundants acting on the released structure.	Determine $[P_\Delta]$, the forces required at the coordinates in the unrestrained structure to cause the independent displacement components $[\Delta]$.
7.	Compute the net displacements at the coordiantes. $[\Delta] = [\Delta_L] + [\Delta_R]$	Compute the net forces at the coordinates. $[P] = [P'] + [P_\Delta]$
8.	Use the condtions of compatiblility of displacements to compute the reduntands. $[P] = [\delta]^{-1}\{[\Delta]-[\Delta_L]\}$	Use the conditions of equilibrium of forces to compute the displacements. $[\Delta] = [k]^{-1}\{[P] - [P']\}$
9.	Knowing the redundants, compute the internal member forces by using equations of statics.	Knowing the displacements, compute the internal member forces by using slope-deflection equations.

Examples:

Determine the degree of static indeterminacy of the pin-jointed plane frame shown in Fig. 1.8.

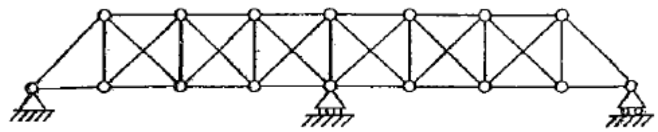


Fig. 1.8

Solution

Total number of independent external reaction components,

$$r = 2 + 1 + 1 = 4$$

Using Eq. (1.7), degree of external indeterminacy,

$$D_{se} = 4 - 3 = 1$$

Number of joints, $j = 16$

Actual number of members, $m = 35$

Using Eq. (1.8), minimum number of members required to preserve geometry of the frame,

$$m' = 2 \times 16 - 3 = 29$$

Using Eq. (1.10), degree of internal indeterminacy,

$$D_{si} = 35 - 29 = 6$$

Hence, degree of static indeterminacy

$$D_s = D_{se} + D_{si} = 1 + 6 = 7$$

Alternatively, the degree of static indeterminacy may be computed using Eq. (1.16). Substituting

$$m = 35 \quad r = 4 \quad j = 16$$

into Eq. (1.16)

$$D_s = 35 + 4 - 2 \times 16 = 7$$

Determine the degree of static indeterminacy of the rigid-jointed plane frame shown in Fig. 1.9.

Solution

Total number of independent external reaction components,

$$r = 2 \times 3 + 2 + 1 = 9$$

Using Eq. (1.7), degree of external indeterminacy,

$$D_{se} = 9 - 3 = 6$$

The number of cuts required to obtain an open configuration, $c = 12$. For instance, cuts may be made in all the beams except in the topmost beams. Using Eq. (1.12), degree of internal indeterminacy

$$D_{si} = 3 \times 12 = 36$$

Hence, degree of static indeterminacy,

$$D_s = D_{se} + D_{si}$$
$$= 6 + 36 = 42$$

Alternatively, the degree of static indeterminacy may be computed using Eq. (1.18). Substituting

$$m = 35$$
$$r = 9$$
$$j = 24$$

into Eq. (1.18),

$$D_s = 3 \times 35 + 9 - 3 \times 24 = 42$$

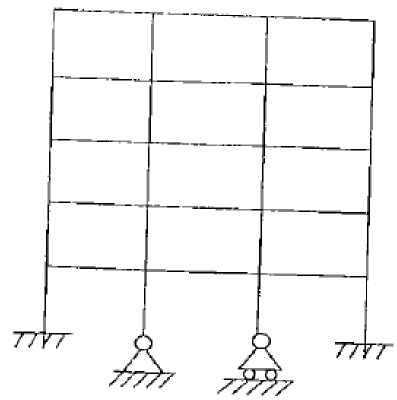


Fig. 1.9

Determine the degree of static indeterminacy of the bow-string girder shown in Fig. 1.10. Assume all joints to be rigid.

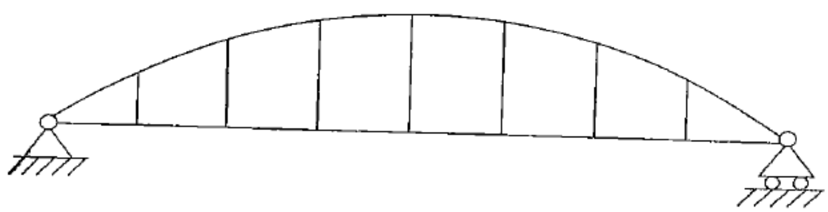


Fig. 1.10

Solution

Total number of independent external reaction components, $r = 3$. Degree of external indeterminacy,

$$D_{se} = 3 - 3 = 0$$

The number of cuts required to obtain an open configuration, $c = 8$. For instance, a cut may be made in the horizontal member in each cell. Using Eq. (1.12), degree of internal indeterminacy,

$$D_{si} = 3 \times 8 = 24$$

Hence, degree of static indeterminacy,

$$D_s = D_{se} + D_{si} = 0 + 24 = 24$$

Alternatively, the degree of static indeterminacy may be computed using Eq. (1.18). Substuting

$$m = 23 \quad r = 3 \quad j = 16$$

into Eq. (1.18),

$$D_s = 3 \times 23 + 3 - 3 \times 16 = 24$$

Determine the degree of static indeterminacy of the rigid-jointed building frame shown in Fig. 1.13(a).

Solution

Total number of independent external reaction components,

$$r = 6 \times 6 = 36$$

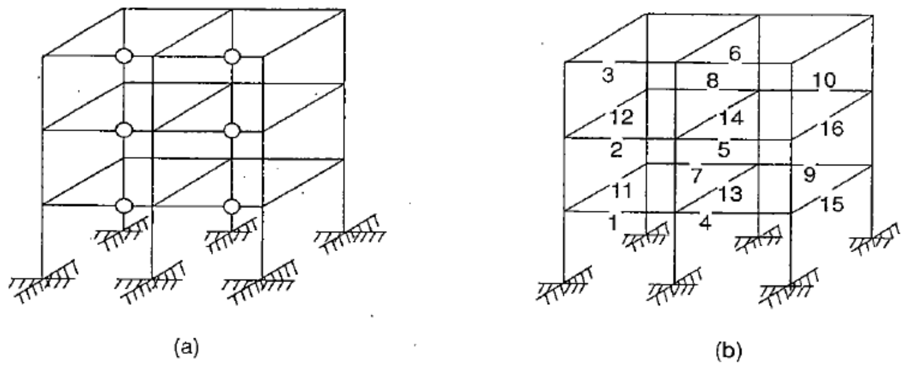


Fig. 1.13

Degree of external indeterminacy,

$$D_{se} = 36 - 6 = 30$$

Number of cuts required to obtain an open configuration, $c = 16$ [Fig. 1.13(b)]. Using Eq. (1.13), degree of internal indeterminacy,

$$D_{si} = 6 \times 16 = 96$$

Hence, degree of static indeterminacy of the frame,

$$D_s = D_{se} + D_{si} = 30 + 96 = 126$$

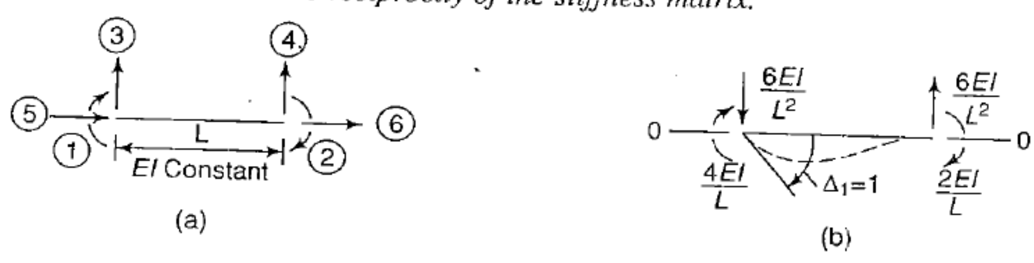
Alternatively, the degree of static indeterminacy may be computed using Eq. (1.19). Substituting

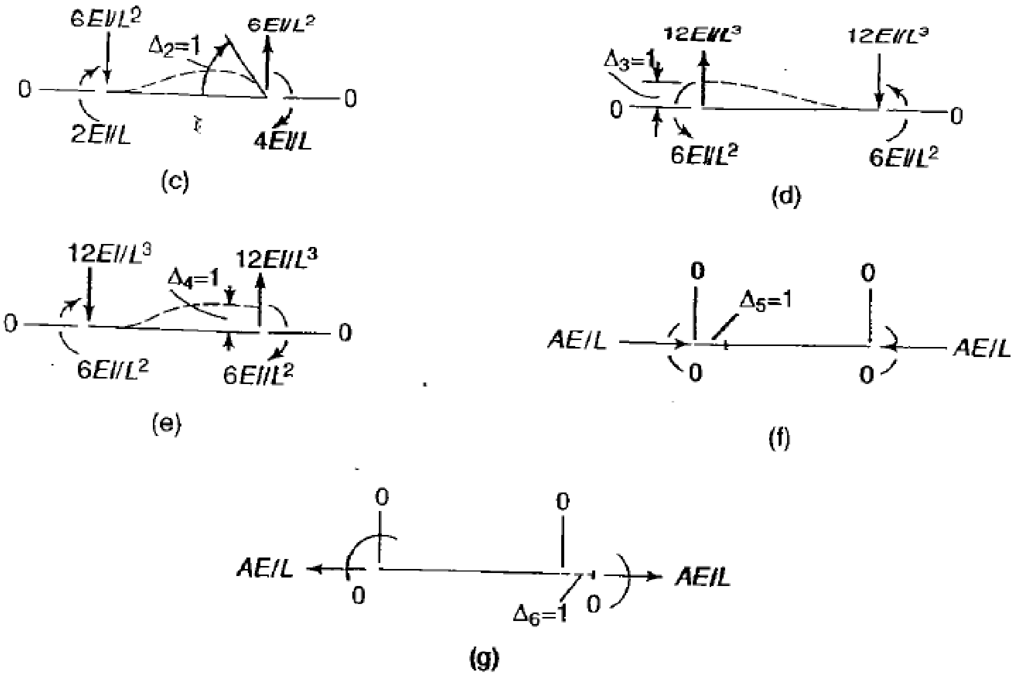
$$m = 39 \quad r = 36 \quad j = 24$$

into Eq. (1.19),

$$D_s = 6 \times 39 + 36 - 6 \times 24 = 126$$

Develop the stiffness matrix for the end-loaded prismatic member AB with reference to the coordinates shown in Fig. 4.4(a). Comment on the relevance of the chosen coordinates. Examine the reciprocity of the stiffness matrix.





The stiffness matrix of the member can be developed by giving a unit displacement successively at each coordinate without any displacement at other coordinates. The forces at coordinates 1 to 6, when a unit displacement is given successively at each of the coordinates 1 to 4, may be computed by using the equations given in Sec. 2.14. For example, when a unit displacement is given at coordinate 1, the forces at coordinates 1 to 6, which constitute the elements of the first column of the stiffnes matrix, are

$$k_{11} = \frac{4EI}{L} \qquad k_{21} = \frac{2EI}{L}$$

$$k_{31} = -\frac{6EI}{L^2} \qquad k_{41} = \frac{6EI}{L^2}$$

$$k_{51} = k_{61} = 0$$

Similarly, the elements of the second, third and fourth columns of the stiffness matrix can be determined.

When a unit displacement is given at coordinate 5 without any displacement at other coordinates, the forces evidently are

$$k_{15} = k_{25} = k_{35} = k_{45} = 0 \qquad k_{55} = \frac{AE}{L} \qquad k_{65} = -\frac{AE}{L}$$

These forces constitute the elements of the fifth column of the stiffness matrix. The sixth column of the stiffness matrix may be generated in a similar manner by giving a unit displacement at coordinate 6.

The deformed shape of the member, when unit displacement is given succesively at coordinates 1 to 6, together with the resulting forces required to sustain the deformed shape of the member, are shown in the free-boy diagrams in Fig. 4.4(b) to (g). Thus the stiffness matrix of member AB with reference to the chosen coordinates may be written as

$$[k] = \begin{bmatrix} \frac{4EI}{L} & \frac{2EI}{L} & -\frac{6EI}{L^2} & \frac{6EI}{L^2} & 0 & 0 \\ \frac{2EI}{L} & \frac{4EI}{L} & -\frac{6EI}{L^2} & \frac{6EI}{L^2} & 0 & 0 \\ -\frac{6EI}{L^2} & -\frac{6EI}{L^2} & \frac{12EI}{L^3} & -\frac{12EI}{L^3} & 0 & 0 \\ \frac{6EI}{L^2} & \frac{6EI}{L^2} & -\frac{12EI}{L^3} & \frac{12EI}{L^3} & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{AE}{L} & -\frac{AE}{L} \\ 0 & 0 & 0 & 0 & -\frac{AE}{L} & \frac{AE}{L} \end{bmatrix} \quad (4.27)$$

where A = area of cross-section of the member
L = length of the member.

Two steel bars AB and BC, each having a cross-sectional area of 20 mm², are connected in series as shown in Fig. 4.10. Develop the flexibility and stiffness matrices with reference to coordinates 1 and 2 shown in the figure. Verify that the two matrices are the invervse of eachy other. Take E = 200 kN/mm².

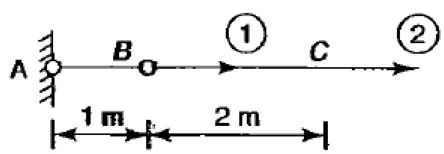


Fig. 4.10

Solution

$$\text{Axial flexibility of bar } AB = \frac{L}{AE} = \frac{1000}{20 \times 200} = 0.25 \text{ mm/kN}$$

$$\text{Axial stiffness of bar } AB = \frac{AE}{L} = 4 \text{ kN/mm}$$

$$\text{Axial flexibility of bar } BC = \frac{L}{AE} = \frac{2000}{20 \times 200} = 0.5 \text{ mm/kN}$$

$$\text{Axial stiffness of bar } BC = \frac{AE}{L} = 2 \text{ kN/mm}$$

The flexibility matrix can be developed by applying a unit force successively at coordinates 1 and 2 and evaluating the displacements at coordinates 1 and 2. To generate the first column of the flexibility matrix, apply a unit force at coordinate 1. The displacements at coordinates 1 and 2 are

$$\delta_{11} = \delta_{21} = 0.25 \text{ mm}$$

Similarly, to generate the second column of the flexibility matrix, apply a unit force at coordinate 2. The displacements at coordinates 1 and 2 are

$$\delta_{12} = 0.25 \text{ mm}$$

$$\delta_{22} = 0.25 + 0.5 = 0.75 \text{ mm}$$

Hence, the required flexibility matrix $[\delta]$ is given by the equation

$$[\delta] = \begin{bmatrix} 0.25 & 0.25 \\ 0.25 & 0.75 \end{bmatrix}$$

The stiffness matrix can be developed by giving a unit displacement successively at coordinates 1 and 2 without any displacement at the other coordinate and determining the forces required at coordinates 1 and 2. To generate the first column of the stiffness matrix, give a unit displacement at coordinate 1. The forces required at coordinates 1 and 2 are

$$k_{11} = 4 + 2 = 6 \text{ kN}$$

$$k_{21} = -2 \text{ kN}$$

To generate the second column of the stiffness matrix, give a unit displacement at coordinate 2. The forces required at coordinates 1 and 2 are

$$k_{12} = -2 \text{ kN}$$

$$k_{22} = 2\text{kN}$$

Hence, the required stiffness matrix $[k]$ is given by the equation

$$[k] = \begin{bmatrix} 6 & -2 \\ -2 & 2 \end{bmatrix}$$

Multiplying the flexibility and stiffness matrices,

$$[\delta][k] = \begin{bmatrix} 0.25 & 0.25 \\ 0.25 & 0.75 \end{bmatrix}\begin{bmatrix} 6 & -2 \\ -2 & 2 \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

As the product of the two matrices is a unit matrix, the two matrices are the inverse of each other.

Develop the flexibility and stiffness matrices for prismatic member AB with reference to the coordinates shown in Fig. 4.11 (a) for the following support conditons:

(i) hinged support at A and roller support at B

(ii) fixed supports at A and B

(iii) fixed support at A and roller support at B.

Verify in each case that the flexibility and stiffness matrices are the inverse of each other.

Solution

(i) The support conditions are shown in Fig. 4.11(b). The flexibility matrix can be developed by applying a unit force successively at coordinates 1 and 2 and evaluating displacements at coordiantes 1 and 2. To generate the first column of the flexibility matrix, apply a unit force at coordinate 1. Using Eqs (A. 71) and (A.72) of Appendix A, the displacement at coordinates 1 and 2 are

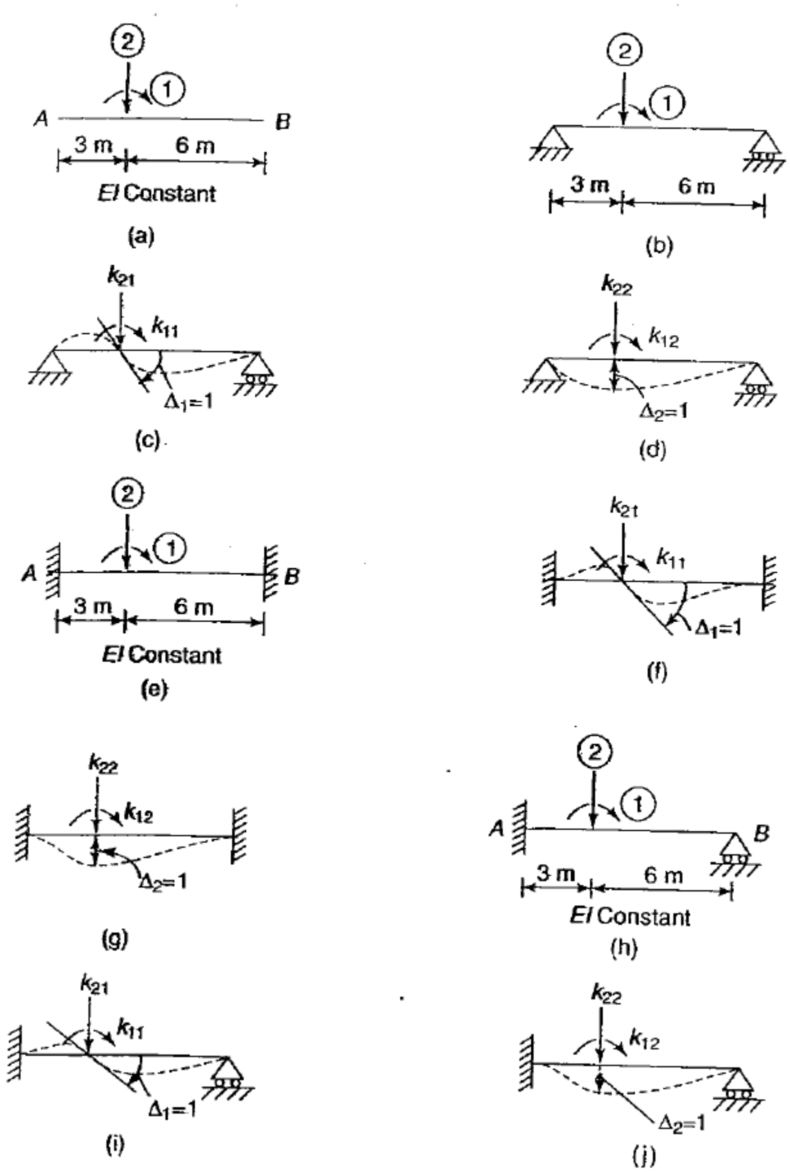


Fig. 4.11

$$\delta_{11} = \frac{1}{3 \times 9EI}[3 \times 3^2 - 3 \times 3 \times 9 + 9^2] = \frac{1}{EI}$$

$$\delta_{21} = \frac{3(9-3)(9-6)}{3 \times 9EI} = \frac{2}{EI}$$

To generate the second column of the flexibility matrix, apply a unit force at coordinate 2. Using Eqs (A.63) and (A.64) of Appendix A, the displacements at coordinates 1 and 2 are

$$\delta_{12} = \frac{3(9-3)(9-6)}{3 \times 9EI} = \frac{2}{EI}$$

$$\delta_{22} = \frac{3^2 \times 6^2}{3 \times 9EI} = \frac{12}{EI}$$

Hence, the required flexibility matrix $[\delta]$ is given by the equation

$$[\delta] = \frac{1}{EI}\begin{bmatrix} 1 & 2 \\ 2 & 12 \end{bmatrix}$$

The stiffness matrix can be developed by giving a unit displacement successively at coordinates 1 and 2 without any displacement at the other coordinate and determining the forces required at coordinates 1 and 2. To generate the first column of the stiffness matrix, give a unit displacement at coordinate 1 as shown in Fig. 4.11(c). The forces required at the coordinates are

$$k_{11} = \frac{3EI}{3} + \frac{3EI}{6} = 1.5EI$$

$$k_{21} = -\frac{3EI}{3^2} + \frac{3EI}{6^2} = -0.25EI$$

To generate the second column of the stiffness matrix, give a unit displacement at coordinate 2 as shown in Fig. 4.11(d). The forces required at coordinates 1 and 2 are

$$k_{12} = -\frac{3EI}{3^2} + \frac{3EI}{6^2} = -0.25EI$$

$$k_{22} = \frac{3EI}{3^3} + \frac{3EI}{6^3} = 0.125EI$$

Hence, the required stiffness matrix $[k]$ is given by the equation

$$[k] = EI\begin{bmatrix} 1.500 & -0.250 \\ -0.250 & 0.125 \end{bmatrix}$$

Multiplying the flexibility and stiffness matrices,

$$[\delta][k] = \frac{1}{EI}\begin{bmatrix} 1 & 2 \\ 2 & 12 \end{bmatrix} EI \begin{bmatrix} 1.500 & -0.250 \\ -0.250 & 0.125 \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

As the product is a unit matrix, the two matrices are the inverse of the each other.

(ii) The support conditions are shown in Fig. 4.11(e). The flexibility matrix can be developed by applying a unit force successively at coordinates 1 and 2 and evaluating the displacement at coordinates 1 and 2. To generate the first column of the flexibility matrix, apply a unit force at coordinate 1. Using Eqs (A.113) and (A.114) of Appendix A, the displacements at coordinates 1 and 2 are

$$\delta_{11} = \frac{3(9-3)(9^2 - 3 \times 3 \times 9 + 3 \times 3^2)}{9^3 EI} = \frac{2}{3EI}$$

$$\delta_{21} = \frac{3^2}{2 \times 9^3 EI} \times (9-3)^2(9-6) = \frac{2}{3EI}$$

To generate the second column of the flexibility matrix, apply a unit force at coordinate 2. Using Eqs. (A.104) and (A.105) of Appendix A, the displacements at coordinates 1 and 2 are

$$\delta_{12} = \frac{3^2}{2 \times 9^3 EI}(9-3)^2(9-6) = \frac{2}{3EI}$$

$$\delta_{22} = \frac{3^3(9-3)^3}{3 \times 9^3\ EI} = \frac{8}{3EI}$$

Hence, the required flexibility matrix $[\delta]$ is given by the equation

$$[\delta] = \frac{2}{3EI}\begin{bmatrix} 1 & 1 \\ 1 & 4 \end{bmatrix}$$

The stiffness matrix can be developed by giving a unit displacement successively at coordinates 1 and 2 without any displacement at the other coordinate and determining the forces required at coordinates 1 and 2. To generate the first column of the stiffness matrix, give a unit displacement at coordinate 1 as shown in Fig. 4.11 (f). The forces required at coordinates 1 and 2 are

$$k_{11} = \frac{4EI}{3} + \frac{4EI}{6} = 2EI$$

$$k_{21} = -\frac{6EI}{3^2} + \frac{6EI}{6^2} = 0.5EI$$

To generate the second column of the stiffness matrix, give a unit displacement at coordinate 2 as shown in Fig. 4.11(g). The forces required at coordinates 1 and 2 are

$$k_{12} = -\frac{6EI}{3^2} + \frac{6EI}{6^2} = -0.5EI$$

$$k_{22} = \frac{12EI}{3^3} + \frac{12EI}{6^3} = 0.5EI$$

Hence, the required stiffness matrix [k] is given by the equation

$$[k] = EI\begin{bmatrix} 2.0 & -0.5 \\ -0.5 & 0.5 \end{bmatrix}$$

Multiplying the flexibility and stiffness matrices,

$$[\delta][k] = \frac{2}{3EI}\begin{bmatrix} 1 & 1 \\ 1 & 4 \end{bmatrix} EI\begin{bmatrix} 2 & -0.5 \\ -0.5 & 0.5 \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

As the product is a unit matrix, the two matrices are the inverse of each other.

(iii) The support conditions are shown in Fig. 4.11(h). The flexibility matrix can be developed by applying a unit force successively at coordinates 1 and 2 and evaluating the displacements at coordinates 1 and 2. To generate the first column of the flexibility matrix, apply a unit force at coordinate 1. Using Eqs (A.35) and (A.36) of Appendix A, the displacements at coordinates 1 and 2 are

$$\delta_{11} = \frac{3}{4 \times 9^3 EI}\left[4 \times 9^3 - 12 \times 9^2 \times 3 + 12 \times 9 \times 3^2 - 3 \times 3^3\right]$$

$$= \frac{11}{12EI}$$

$$\delta_{21} = \frac{3^2}{4 \times 9^3 EI}[2 \times 9^3 - 6 \times 9^2 \times 3 + 5 \times 9 \times 3^2 - 3^3]$$

$$= \frac{7}{6EI}$$

To generate the second column of the flexibility matrix, apply a unit force at coordinate 2. Using Eqs (A.30) and (A.31) of Appendix A, the displacements at coordinates 1 and 2 are

$$\delta_{12} = \frac{3^2}{4 \times 9^3 EI}\left[2 \times 9^3 - 6 \times 9^2 \times 3 + 5 \times 9 \times 3^2 - 3^3\right]$$

$$= \frac{7}{6EI}$$

$$\delta_{22} = \frac{3^3}{12 \times 9^3 EI}\left[4 \times 9^3 - 9 \times 9^2 \times 3 + 6 \times 9 \times 3^2 - 3^3\right]$$

$$= \frac{11}{3EI}$$

Hence, the required flexibility matrix $[\delta]$ is given by the equation

$$[\delta] = \frac{1}{12EI}\begin{bmatrix} 11 & 14 \\ 14 & 44 \end{bmatrix}$$

The stiffness matrix can be developed by giving a unit displacement successively at coordinates 1 and 2 without any displacement at the other coordinate and determining the forces required at coordinates 1 and 2. To generate the first column of the stiffness matrix, give a unit displacement at coordinate 1 as shown in Fig. 4.11(i). The forces required at coordinates 1 and 2 are

$$k_{11} = \frac{4EI}{3} + \frac{3EI}{6} = \frac{11EI}{6}$$

$$k_{21} = \frac{-6EI}{3^2} + \frac{3EI}{6^2} = \frac{-7EI}{12}$$

To generate the second column of the stiffness matrix, give a unit displacement at coordinate 2 as shown in Fig. 4.11(j). The forces required at coordinates 1 and 2 are

$$k_{12} = -\frac{6EI}{3^2} + \frac{3EI}{6^2} = \frac{-7EI}{12}$$

$$k_{22} = \frac{12EI}{3^3} + \frac{3EI}{6^3} = \frac{11EI}{24}$$

Hence, the required stiffness matrix $[k]$ is given by the equation

$$[k] = \frac{EI}{24}\begin{bmatrix} 44 & -14 \\ -14 & 11 \end{bmatrix}$$

Multiplying the flexibility and stiffness matrices,

$$[\delta][k] = \frac{1}{12EI}\begin{bmatrix} 11 & 14 \\ 14 & 44 \end{bmatrix}\frac{EI}{24}\begin{bmatrix} 44 & -14 \\ -14 & 11 \end{bmatrix} = \begin{bmatrix} 1 & 0 \\ 0 & 1 \end{bmatrix}$$

Develop the flexibility and stiffness matrices for beam AB with reference to the coordinates shown in Fig. 4.12(a).

Solution

The flexibility matrix can be developed by applying a unit force successively at the coordinates and evaluating the displacements at all the coordianates. To generate the first column of the flexibility matrix, apply a unit force at coordinate 1. Using Eqs (A.14), (A.15) and (A.16) of Appendix A, the displacements at the coordinates are

$$\delta_{11} = \frac{10}{EI}$$

$$\delta_{21} = \frac{10 \times 10}{2EI} = \frac{50}{EI}$$

$$\delta_{31} = \frac{10}{EI}$$

$$\delta_{41} = \frac{10(2 \times 20 - 10)}{6EI} = \frac{150}{EI}$$

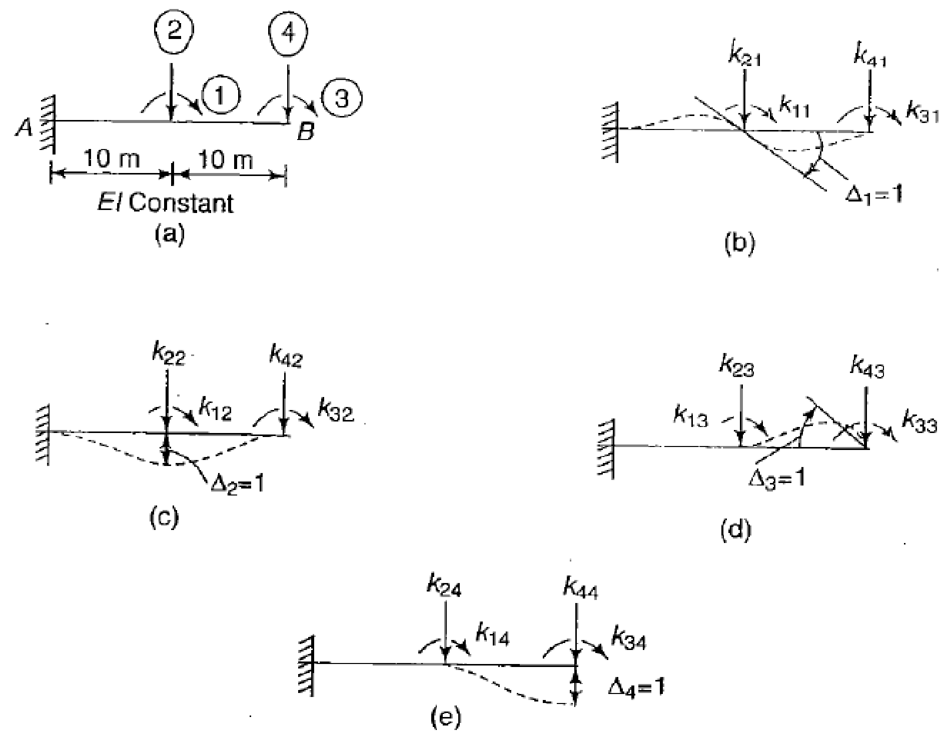


Fig. 4.12

To generate the second column of the flexibility matrix, aply a unit force at coordinate 2. Using Eqs. (A.9), (A.10) and (A.11) of Appendix A, the displacements at the coordinates are

$$\delta_{12} = \frac{10 \times 10}{2EI} = \frac{50}{EI}$$

$$\delta_{22} = \frac{10^3}{3EI} = \frac{1000}{3EI}$$

$$\delta_{32} = \frac{10 \times 10}{2EI} = \frac{50}{EI}$$

$$\delta_{42} = \frac{10^2(3 \times 20 - 10)}{6EI} = \frac{2500}{3EI}$$

To generate the third column of the flexibility matrix, apply a unit force at coordinate 3. Using Eqs (A.5) to (A.8) of Appendix A, the displacements at the coordinates are

$$\delta_{13} = \frac{10}{EI} \qquad \delta_{23} = \frac{10^2}{2EI} = \frac{50}{EI}$$

$$\delta_{33} = \frac{20}{EI} \qquad \delta_{43} = \frac{20^2}{2EI} = \frac{200}{EI}$$

To generate the fourth column of the flexibility matrix, apply a unit force at coordinate 4. Using Eqs (A.1) to (A.4) of Appendix A, the displacements at the coordinates are

$$\delta_{14} = \frac{10(2 \times 20 - 10)}{2EI} = \frac{150}{EI}$$

$$\delta_{24} = \frac{10^2(3 \times 20 - 10)}{6EI} = \frac{2500}{3EI}$$

$$\delta_{34} = \frac{20^2}{2EI} = \frac{200}{EI}$$

$$\delta_{44} = \frac{20^3}{3EI} = \frac{8000}{3EI}$$

Hence, the required flexibility matrix $[\delta]$ is given by equation

$$[\delta] = \frac{1}{3EI}\begin{bmatrix} 30 & 150 & 30 & 450 \\ 150 & 1000 & 150 & 2500 \\ 30 & 150 & 60 & 600 \\ 450 & 2500 & 600 & 8000 \end{bmatrix}$$

The stiffness matrix can be developed by giving a unit displacement successively at each coordinate without any displacement at the other coordinates and determining the forces required at all the coordinates. To generate the first column of the stiffness matrix, give a unit displacement at coordinate 1 as shown in Fig. 4.12(b). The forces required at the coordinates are

$$k_{11} = \frac{4EI}{10} + \frac{4EI}{10} = 0.8EI$$

$$k_{21} = \frac{6EI}{10^2} - \frac{6EI}{10^2} = 0$$

$$k_{31} = \frac{2EI}{10} = 0.2EI$$

$$k_{41} = -\frac{6EI}{10^2} = -0.06EI$$

To generate the second column of the stiffness matrix, give a unit displacement at coordinate 2 as shown in Fig. 4.12(c). The forces required at the coordinates are

$$k_{12} = \frac{6EI}{10^2} - \frac{6EI}{10^2} = 0$$

$$k_{22} = \frac{12EI}{10^3} + \frac{12EI}{10^3} = 0.024EI$$

$$k_{32} = \frac{6EI}{10^2} = 0.06EI$$

$$k_{42} = -\frac{12EI}{10^3} = -0.012EI$$

To generate the third column of the stiffness matrix, give a unit displacement at coordinate 3 as shown in Fig. 4.12(d). The forces required at the coordinates are

$$k_{13} = \frac{2EI}{10} = 0.2EI$$

$$k_{23} = \frac{6EI}{10^2} = 0.06EI$$

$$k_{33} = \frac{4EI}{10} = 0.4EI$$

$$k_{43} = \frac{-6EI}{10^2} = -0.06EI$$

To generate the fourth column of the stiffness matrix, give a unit displacement at coordinate 4 as shown in Fig. 4.12(e). The forces required at the coordinates are

$$k_{14} = \frac{-6EI}{10^2} = -0.06EI$$

$$k_{24} = \frac{-12EI}{10^3} = -0.012EI$$

$$k_{34} = \frac{-6EI}{10^2} = -0.06EI$$

$$k_{44} = \frac{12EI}{10^3} = 0.012EI$$

Hence, the required stiffness matrix $[k]$ is given by the equation

$$[k] = EI \begin{bmatrix} 0.800 & 0 & 0.200 & -0.060 \\ 0 & 0.024 & 0.060 & -0.012 \\ 0.200 & 0.060 & 0.400 & -0.060 \\ -0.060 & -0.012 & -0.060 & 0.012 \end{bmatrix}$$

In this example the computational effort required for developing the flexibility matrix is approximately the same as that for the stiffness matrix.

Analysis of pin-jointed frames by Stiffness Matrix method

Unit displacement in coordinate direction j:

Consider the Figure 11.48.

$$AA' = 1$$

Therefore, the shortening of member $AB = AA' \sin\theta = \sin\theta$

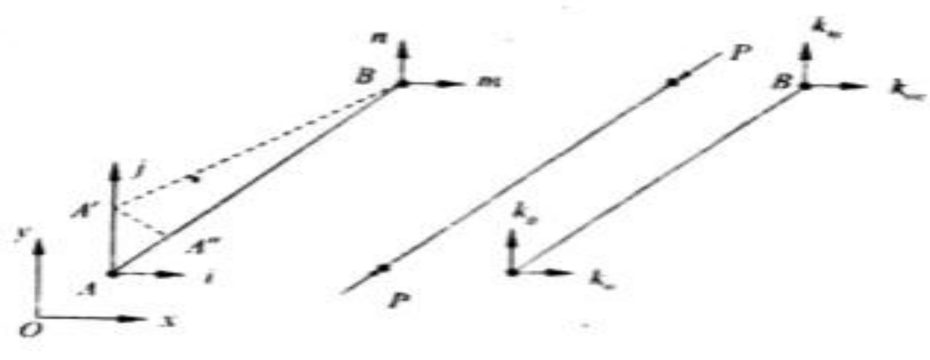


Figure 11.48: Unit displacement in coordinate direction j.

Therefore, the axial compressive force P developed is given by

$$\frac{PL}{AE} = \sin\theta$$

or
$$P = \frac{AE}{L}\sin\theta$$

$\therefore$
$$k_{ij} = P\cos\theta = \frac{AE}{L}\times\cos\theta\sin\theta$$

$$k_{ji} = P\sin\theta = \frac{AE}{L}\times\sin^2\theta$$

$$k_{mi} = -P\cos\theta = -\frac{AE}{L}\times\sin\theta\cos\theta$$

$$k_{ni} = -P\sin\theta = -\frac{AE}{L}\times\sin^2\theta$$

Joint stiffness will be

$$k_{ij} = \sum\left[\frac{AE}{L}\times\cos\theta\sin\theta\right]$$

$$k_{ji} = \sum\left[\frac{AE}{L}\times\sin^2\theta\right]$$

$$k_{mj} = -\left[\frac{AE}{L}\times\sin\theta\cos\theta\right]$$

$$k_{nj} = -\left[\frac{AE}{L}\times\sin^2\theta\right]$$

Member Forces

Let the final position of member AB be $A'\,B'$ as shown in Figure 11.49. Note that, for deriving the expression, $A'\,B'$ is selected such that all the displacements are positive.

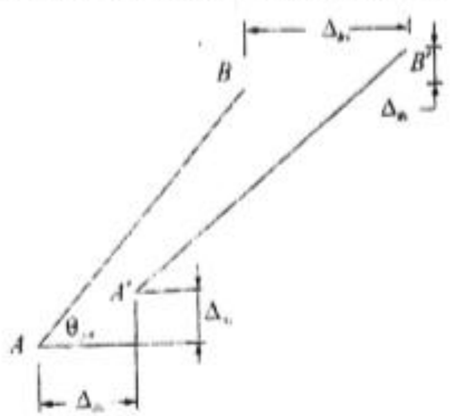


Figure 11.49: Final position of member AB.

Shortening of member due to displacement at A

$$= \Delta_{AX} \cos \theta_{AB} + \Delta_{AY} \sin \theta_{AB}$$

Extension of the member due to displacement at B

$$= \Delta_{BX} \cos \theta_{AB} + \Delta_{BY} \sin \theta_{AB}$$

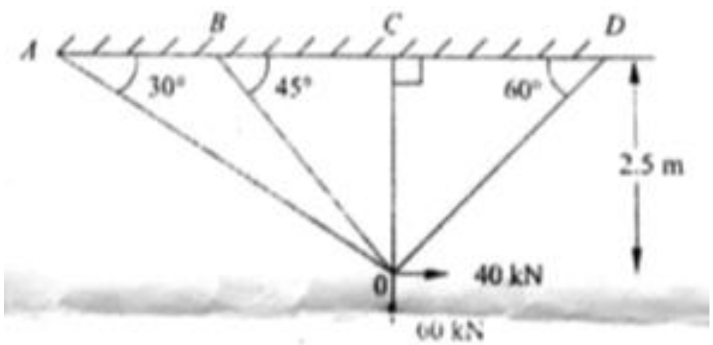
Therefore, the extension of member AB

$$= (\Delta_{BX} - \Delta_{AX}) \cos \theta_{AB} + (\Delta_{BY} - \Delta_{AY}) \sin \theta_{AB}$$

$$\therefore \qquad P_{AB} = \frac{AE}{L}[(\Delta_{BX} - \Delta_{AX}) \cos \theta_{AB} + (\Delta_{BY} - \Delta_{AY}) \sin \theta_{AB}]$$

Example :

Analysethe pin-jointed truss as shown in figure by stiffness matrix method. Take area od cross-section for all members = 1000 mm^2 and modulus of elasticity E = 200 kN/mm^2



Solution Degree of freedom = 2

The coordinates are selected as shown in Figure 11.50(*b*). Table 11.4 is prepared.

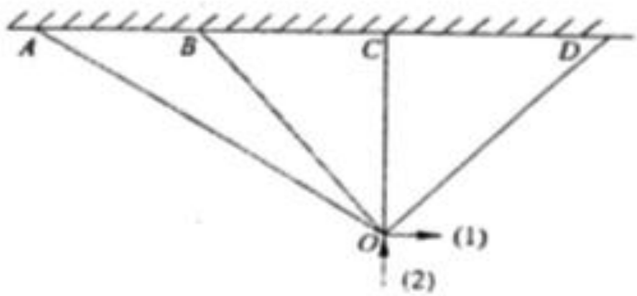


Figure 11.50(b): Coordinates selected.

Table 11.4: Calculations for assembling stiffness

Member	$\frac{AE}{L}$	θ	$\frac{AE}{L}\cos^2\theta$	$\frac{AE}{L}\cos\theta\sin\theta$	$\frac{AE}{L}\sin^2\theta$
OA	40	150°	30	−17.321	10
OB	56.569	135°	28.285	−28.285	28.285
OC	80.0	90°	0	0	80.000
OD	69.282	60°	17.321	30	51.962
		Σ	75.606	−15.606	170.247

$$k_{11} = \sum \frac{AE}{L}\cos^2\theta = 75.606$$

$$k_{21} = k_{12} = \sum\left[\frac{AE}{L}\times\cos\theta\sin\theta\right] = -15.606$$

$$k_{22} = \sum\left[\frac{AE}{L}\times\sin^2\theta\right] = 170.247$$

$$\begin{bmatrix} P_1 \\ P_2 \end{bmatrix} = \begin{bmatrix} 40 \\ -60 \end{bmatrix}$$

Therefore, the stiffness equation is

$$\begin{bmatrix} \Delta_1 \\ \Delta_2 \end{bmatrix} = \left(\frac{1}{12628.12}\right)\begin{bmatrix} 170.247 & 15.606 \\ 15.606 & 75.606 \end{bmatrix}\begin{bmatrix} 40 \\ -60 \end{bmatrix}$$

$$= \begin{bmatrix} 0.465 \\ -0.310 \end{bmatrix}$$

$$P_{OA} = \frac{AE}{L}\left[(\Delta_{OX} - \Delta_{AX})\cos\theta_{OA} + (\Delta_{OY} - \Delta_{AY})\sin\theta_{OA}\right]$$

$= 40\ [(0 - 0.465)\cos 150° + (0 + 0.310)\sin 150°]$

$= 22.308$ kN

$P_{OB} = 56.569\ [(0 - 0.456)\cos 135° + (0 + 0.310)\sin 135°]$

$= 31.000$ kN

$P_{OC} = 80\ [(0 - 0.465)\cos 90° + (0 + 0.310)\sin 90°]$

$= 24.8$ kN

$P_{OD} = 69.282\ [(0 - 0.465)\cos 60° + (0 + 0.310)\sin 60°]$

$= 2.492$ kN

UNIT - IV

BOUNDARY VALUE PROBLEMS (BVP)

The given indeterminate structure is first made kinematically determinate by introducing constraints atthenodes. The required number of constraints is equal to degrees of freedom at the nodes that is kinematic indeterminacy k. The kinematic ally determinate structure comprises of fixed ended members, hence, all nodal displacements are zero. These results in stress resultant discontinuities at these nodes under the action of applied loads or in other words the clamped joints are not in equilibrium.Inorder to restore the equilibrium of stress resultants at the nodes the nodes are imparted suitable unknown displacements. The number of simultaneous equations representing joint equilibrium of forces is equal to kinematic indeterminacy k. Solution of these equations gives unknown nodal displacements. Using stiffness properties of members the memberend forces are computed and hence the internal forces throughout the structure.

Since nodal displacements are unknowns, the method is also called displacement method. Since equilibrium conditions are applied at the joints the method is also called equilibrium method. Since stiffness properties of members are used the method is also called stiffness method.In the displacement method of analysis the equilibrium equations are written by expressing the unknown joint displacements in terms of loads by using load-displacement relations. The unknown joint displacements (the degrees of freedom of the structure) are calculated by solving equilibrium equations. The slope -deflection and moment - distribution methods were extensively used before the high speed computing era. After the revolution in computer industry, only direct stiffness method is used.

PROPERTIES OFTHESTIFFNESS MATRIX

The properties of the stiffness matrix are:

- It is asymmetric matrix
- The sum of elements in any column must be equal to zero.
- It is an unstable element therefore the determinant is equal to zero.

ELEMENT AND GLOBAL STIFFNESS MATRICES

Local co ordinates

In the analysis for convenience we fix the element coordinates coincident with the member axis called element (or) local coordinates (coordinates defined along the individual member axis)

Global co ordinates

It is normally necessary to define a coordinate system dealing with the entire structure is called system on global coordinates (Common coordinate system dealing with the entire structure)

Transformation matrix

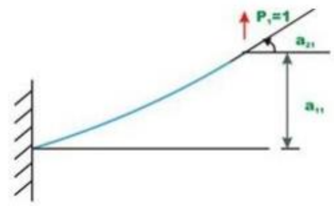
The connectivity matrix which relates the internal forces Q and the external forces R is known as the force transformation matrix. Writing it in a matrix form,

$$\{Q\} = [b]\{R\}$$

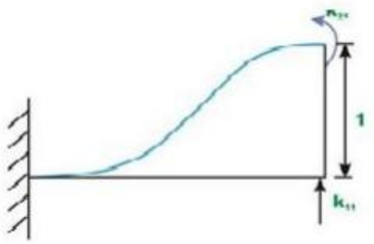
Where Q=member force matrix/vector, b=force transformation matrix R = external force/load matrix/ vector

ANALYSIS OF CONTINUOUS BEAMS

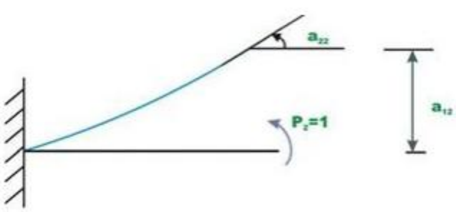
Cantilever Beam



Cantilever Beam with unit load along P_1



Cantilever Beam with unit Displacement along U_1



Cantilever Beam with unit Moment along P_2

ANALYSIS OF PIN JOINTED PLANE FRAMES

An introduction to the stiffness method was given in the previous Page. The basic principles involved in the analysis of beams, trusses were discussed. The problems were solved with hand computation by the direct application of the basic principles. The procedure discussed in the session (page) though enlighteningarenot suitable for computer programming. It is necessary to keeps hand computation to a minimum while implementing this procedure on the computer.

In this session a formal approach has been discussed which may be readily programmed on a computer. In this less on the direct stiffness method as applied to planar truss structure is discussed.

Planetrusses are made up of short thin members inter connected a thin goesto form triangulated patterns. Ahinge connection can only transmit forces from one member to another member but not the moment. For analysis purpose, the truss is loaded at the joints. Hence, a truss member is subjected to only axial forces and the forces remain constant along the length of the member. The forces in the member at its two ends must be of the same magnitude but actin the opposite directions for equilibrium.

The finite element analysis is a numerical technique. In this method all the complexities of the problems, likevarying shape, boundary conditions and loads are maintained as they are but the solutions obtained areapproximate. Because of its diversity and flexibility as an analysis tool, it is receiving much attention inengineering. The fast improvements in computer hardware technology and slashing of cost of computershave boosted this method, since the computer is the basic need for the application of this method. A numberof popular brand of finite element analysis packages are now available commercially. Some of the popularpackages are STAAD-PRO, GT-STRUDEL, NASTRAN, NISA and ANSYS. Using these packages one can analyze several complex structures.

The finite element analysis originated as a method of stress analysis in the design of aircrafts. It started asan extension of matrix method of structural analysis. Today this method is used not only for the analysis insolid mechanics, but even in the analysis of fluid flow, heat transfer, electric and magnetic fields and manyothers. Civil engineers use this method extensively for the analysis of beams, space frames, plates, shells,folded plates, foundations, rock mechanics problems and seepage analysis of fluid through porous media.Both static and dynamic problems can be handled by finite element analysis. This method is used extensivelyfor the analysis and design of ships, aircrafts, space crafts, electric motors and heat engines.

The various steps involved in the finite element analysis are:

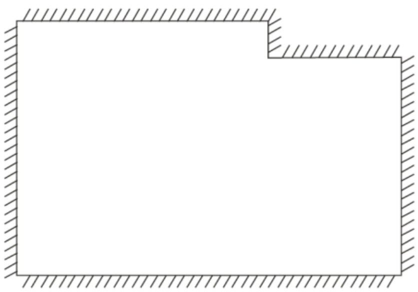
i. Select suitable field variables and the elements.
ii. Discretize the continua.
iii. Select interpolation functions.
iv. Find the element properties.
v. Assemble element properties to get global properties.
vi. Impose the boundary conditions.
vii. Solve the system equations to get the nodal unknowns.
viii. Make the additional calculations to get the required values.

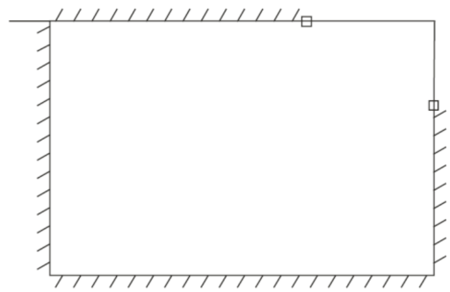
FINITE ELEMENT METHOD VS CLASSICAL METHODS

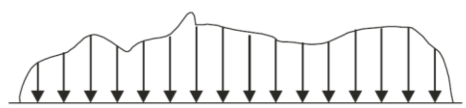
1. In classical methods exact equations are formed and exact solutions are obtained where as in finiteelement analysis exact equations are formed but approximate solutions are obtained.
2. Solutions have been obtained for few standard cases by classical methods, where as solutions canbe obtained for all problems by finite element analysis.
3. Whenever the following complexities are faced, classical method makes the drastic assumptionsand looks for the solutions:
(a) Shape
(b) Boundary conditions
(c) Loading
Fig. shows such cases in the analysis of slabs (plates).
To get the solution in the above cases, rectangular shapes, same boundary condition along a sideand regular equivalent loads are to be assumed. In FEM no such assumptions are made. The problemis treated as it is.
4. When material property is not isotropic, solutions for the problems become very difficult in classicalmethod. Only few simple cases have been tried successfully by researchers. FEM can handlestructures with anisotropic properties also without any difficulty.
5. If structure consists of more than one material, it is difficult to use classical method, but finiteelement can be used without any difficulty.

6. Problems with material and geometric non-linearities cannot be handled by classical methods. There is no difficulty in FEM.

Hence FEM is superior to the classical methods only for the problems involving a number of complexitieswhich cannot be handled by classical methods without making drastic assumptions. For all regular problems,the solutions by classical methods are the best solutions. Infact, to check the validity of the FEM programsdeveloped, the FEM solutions are compared with the solutions by classical methods for standard problems.


(a) Irregular shaper


(b) Irregular boundary condition


(c) Irregular loading

FEM Vs Finite Diffference Method (FDM)

1. FDM makes pointwise approximation to the governing equations i.e. it ensures continuity only atthe node points. Continuity along the sides of grid lines are not ensured.

FEM make piecewise approximation i.e. it ensures the continuity at node points as well as alongthe sides of the element.

2. FDM do not give the values at any point except at node points. It do not give any approximatingfunction to evaluate the basic values (deflections, in case of solid mechanics) using the nodalvalues.

FEM can give the values at any point. However the values obtained at points other than nodesare by using suitable interpolation formulae.

3. FDM makes stair type approximation to sloping and curved boundaries as shown in figure. FEM can consider the sloping boundaries exactly. If curved elements are used, even the curvedboundaries can be handled exactly.

4. FDM needs larger number of nodes to get good results while FEM needs fewer nodes.

5. With FDM fairly complicated problems can be handled whereas FEM can handle all complicatedproblems.

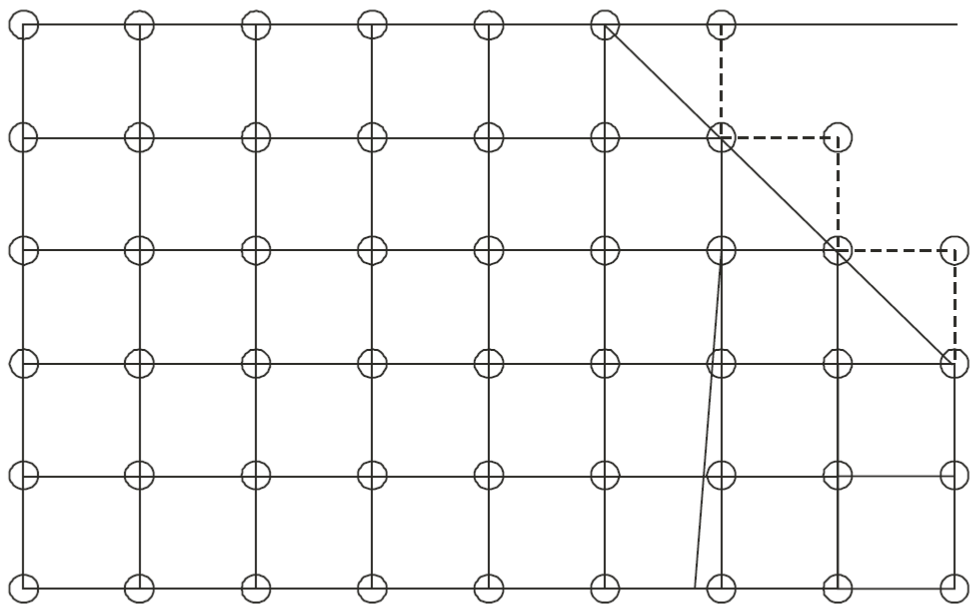


Fig.: FDM approximation of shape

Need for studying FEM

Now, a number of users friendly packages are available in the market. Hence one may ask the question 'Whatis the need to study FEA?'.The above argument is not sound. The finite element knowledge makes a good engineer better while justuser without the knowledge of FEA may produce more dangerous results. To use the FEA packages properly,

The user must know the following points clearly:

1. Which elements are to be used for solving the problem in hand.
2. How to discritise to get good results.
3. How to introduce boundary conditions properly.
4. How the element properties are developed and what are their limitations.
5. How the displays are developed in pre and post processor to understand their limitations.
6. To understand the difficulties involved in the development of FEA programs and hence the need

for checking the commercially available packages with the results of standard cases.

Unless user has the background of FEA, he may produce worst results and may go with overconfidence.Hence it is necessary that the users of FEA package should have sound knowledge of FEA.

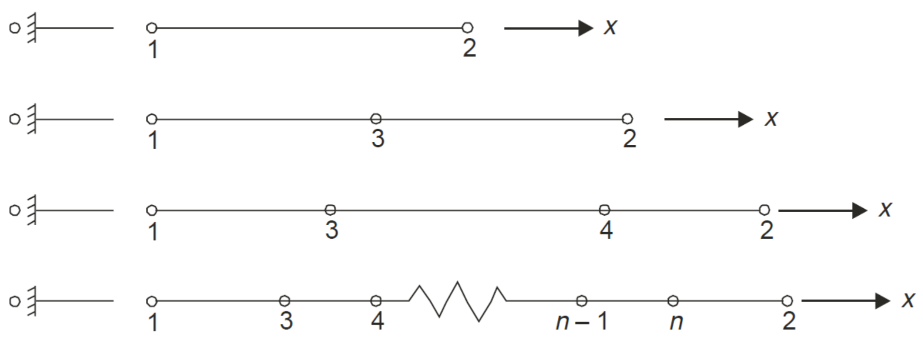
Element Shapes

Based on the shapes elements can be classified as

I. One dimensional elements
II. Two dimensional elements
III. Axi-symmetric elements and
IV. Three dimensional elements.

One dimensional element

These elements are suitable for the analysis of one dimensional problem and may be called as line elementsalso. Figure shows different types of one dimensional elements.

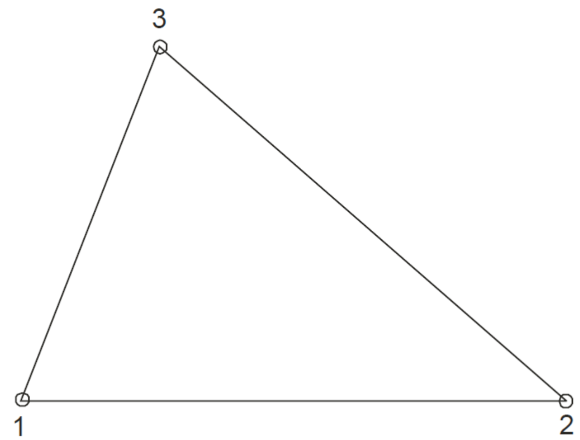


Two dimensional elements

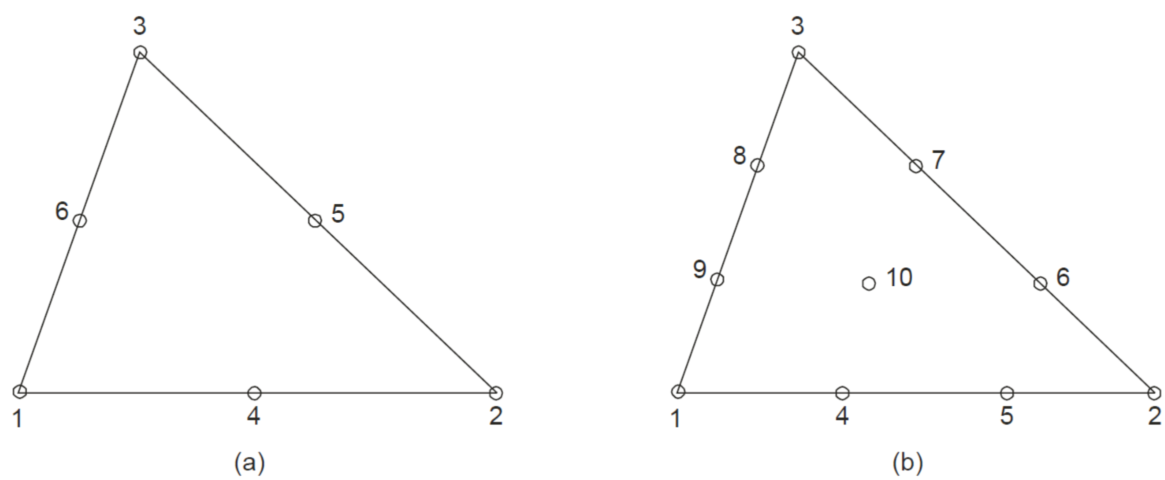
We need two dimensional elements to solve two dimensional problems. Common two dimensional problemsin stress analysis are plane stress, plane strain and plate problems. Two dimensional elements often used isthree noded triangular element shown in figure It has the distinction of being the first and most usedelement. These elements are known as Constant Strain Triangles (CST) or Linear Displacement Triangles.

Six noded and ten noded triangular elements (shown in figure) are also used by the analysts. Six noded triangularelement is known as Linear Strain Triangle (LST) or as Quadratic Displacement Triangle. Ten noded triangular elements are known as Quadratic Strain Triangles (QST) or Cubic Displacement Triangles. Onecan think of trying the use of still higher order triangular elements like Cubic Strain Triangles and QuarticStrain Triangles.

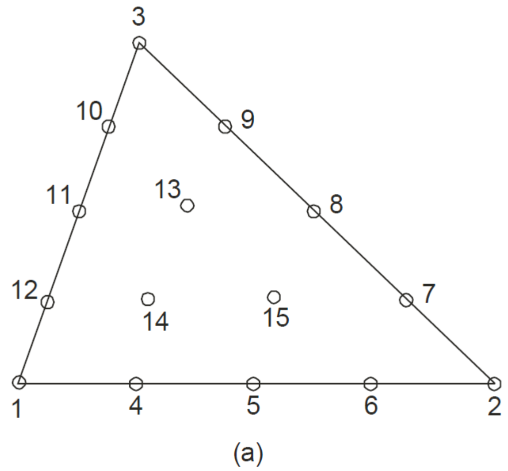
A simple but less used two dimensional element is the four noded rectangular element whose sides areparallel to the global coordinate systems (shown in figure). This systems is easy to construct automatically but it is notwell suited to approximate inclined boundaries.

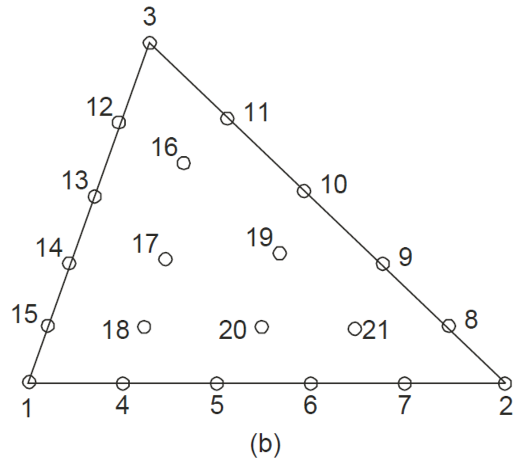


Constant strain triangle



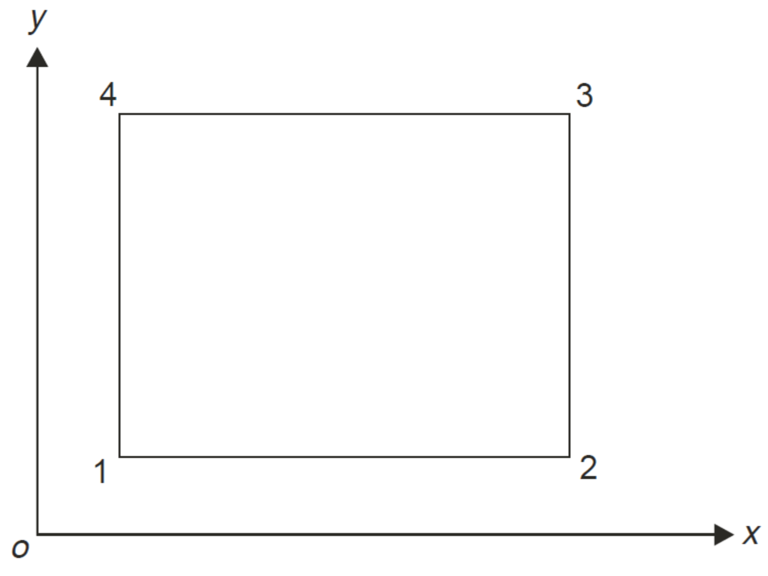
(a) Linear strain triangle (b) Quadratic strain triangle



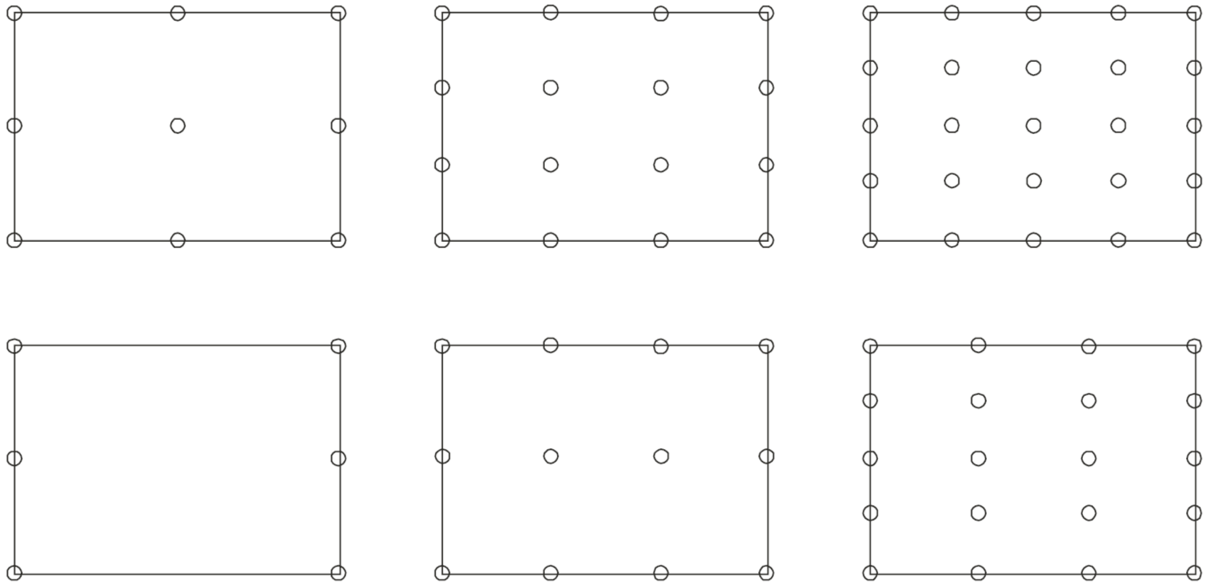


(a) Cubic strain triangle (15 noded) (b) Quartic strain triangle (21 noded)

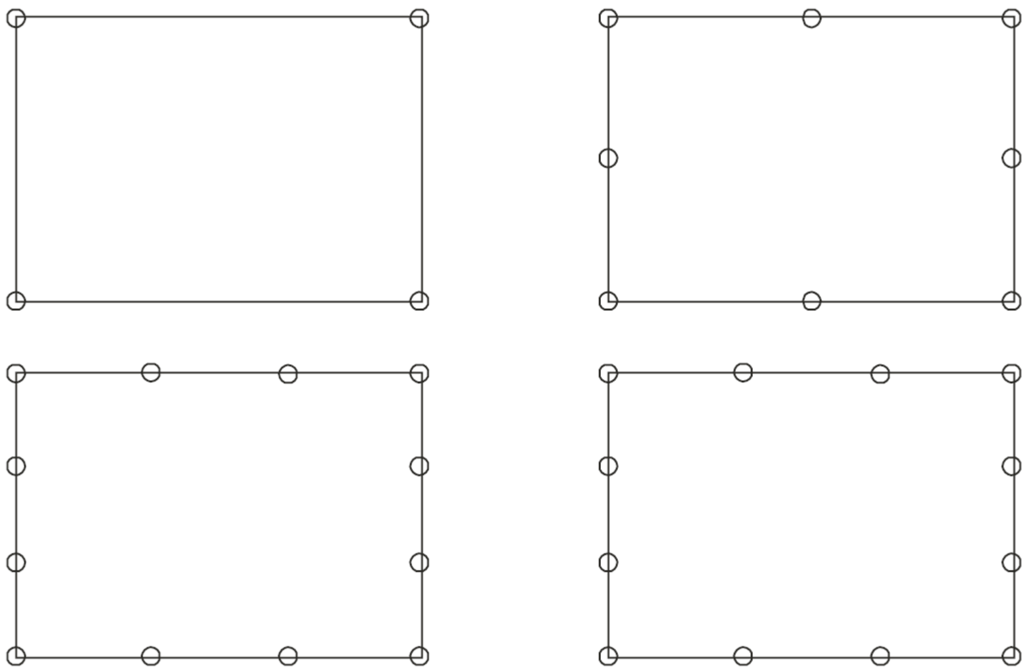
Rectangular elements of higher order also can be used. Figure shows a family of Lagrange rectanglein which nodes are in the form of grid points. Figure shows the family of Serendipity rectangles whichare having nodes only along the external boundaries.



4 noded rectangular element

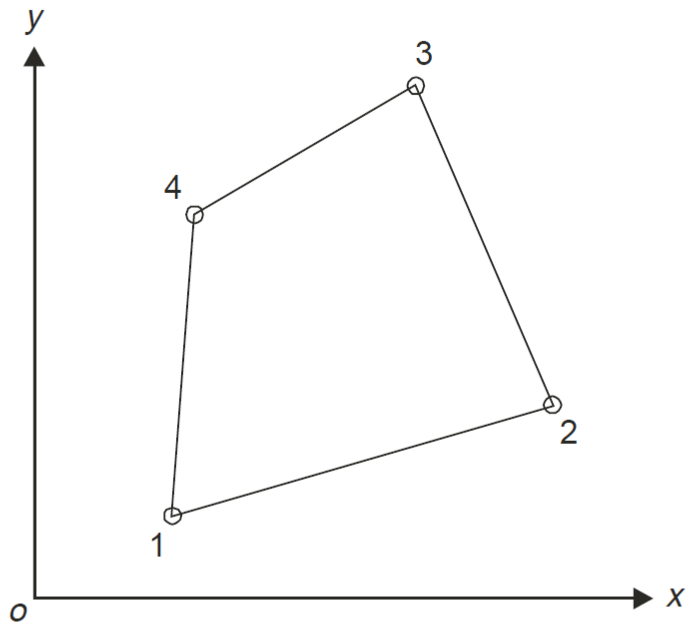


Lagrange family rectangular elements

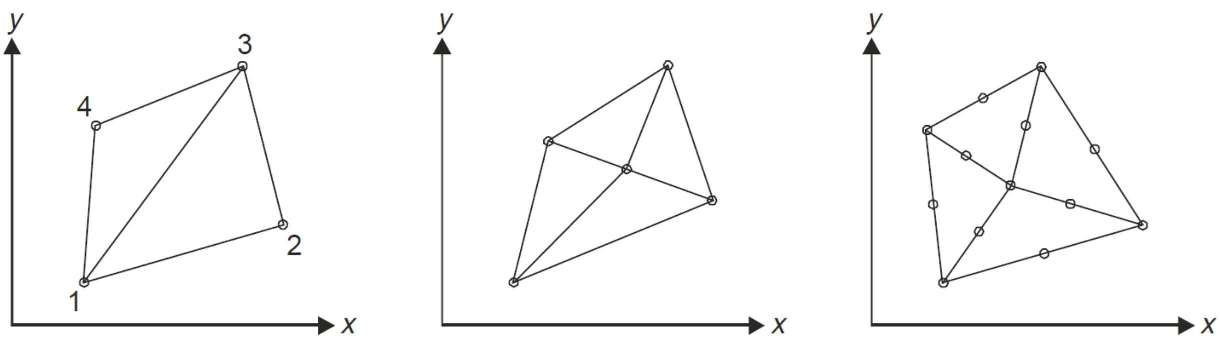


Serendipity family rectangular elements

Quadrilateral Elements are also used in finite element analysis shown in figure. Initially quadrilateral elementswere developed by combining triangular elements shown in figure. But it has taken back stage after isoparametricconcept was developed. Isoparametric concept is based on using same functions for defining geometries andnodal unknowns. Even higher order triangular elements may be used to generate quadrilateral elements.

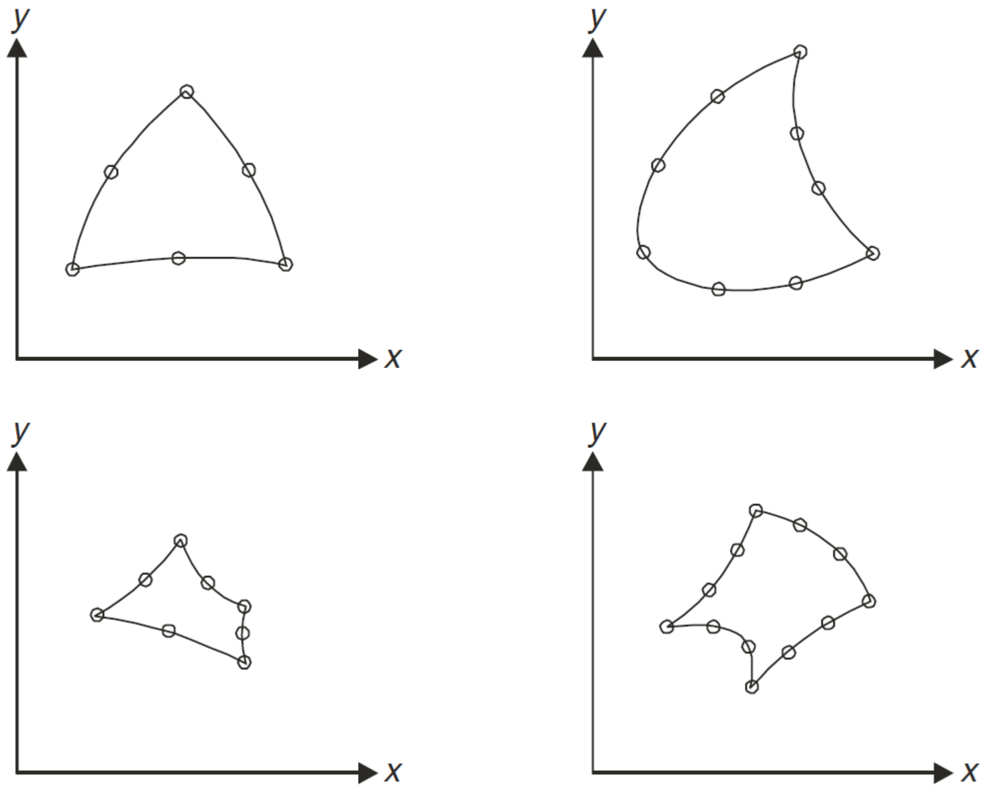


Quardilateral element



Quardilateral elements generated using triangular elements

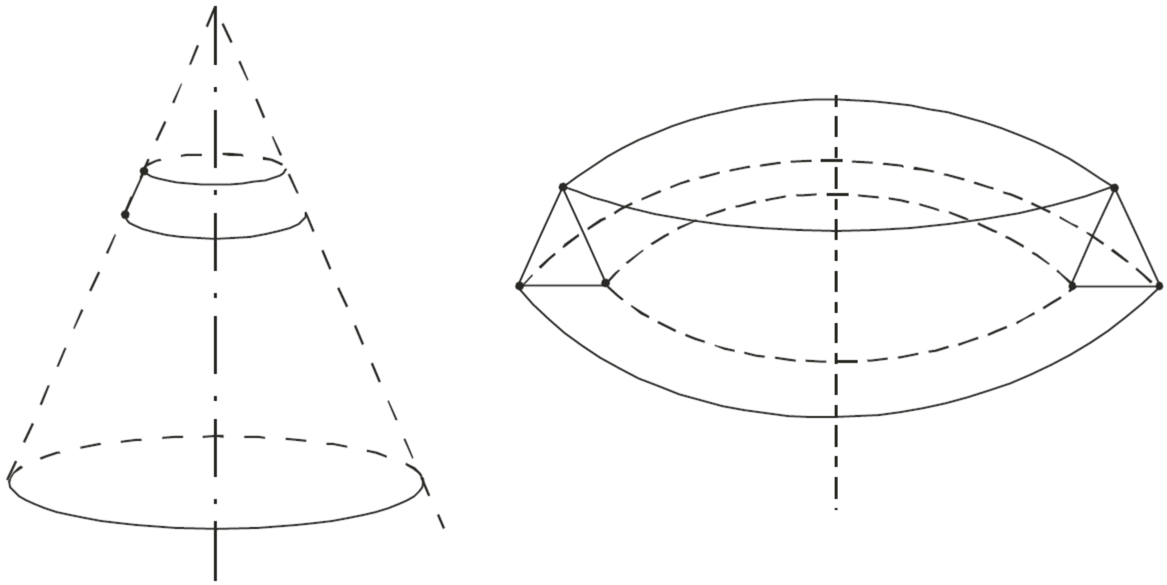
Using isoparametric concept even curved elements are developed to take care of boundaries with curved shapes shown inn below figure.



Curved two dimensional elements

Axi-symmetric elements

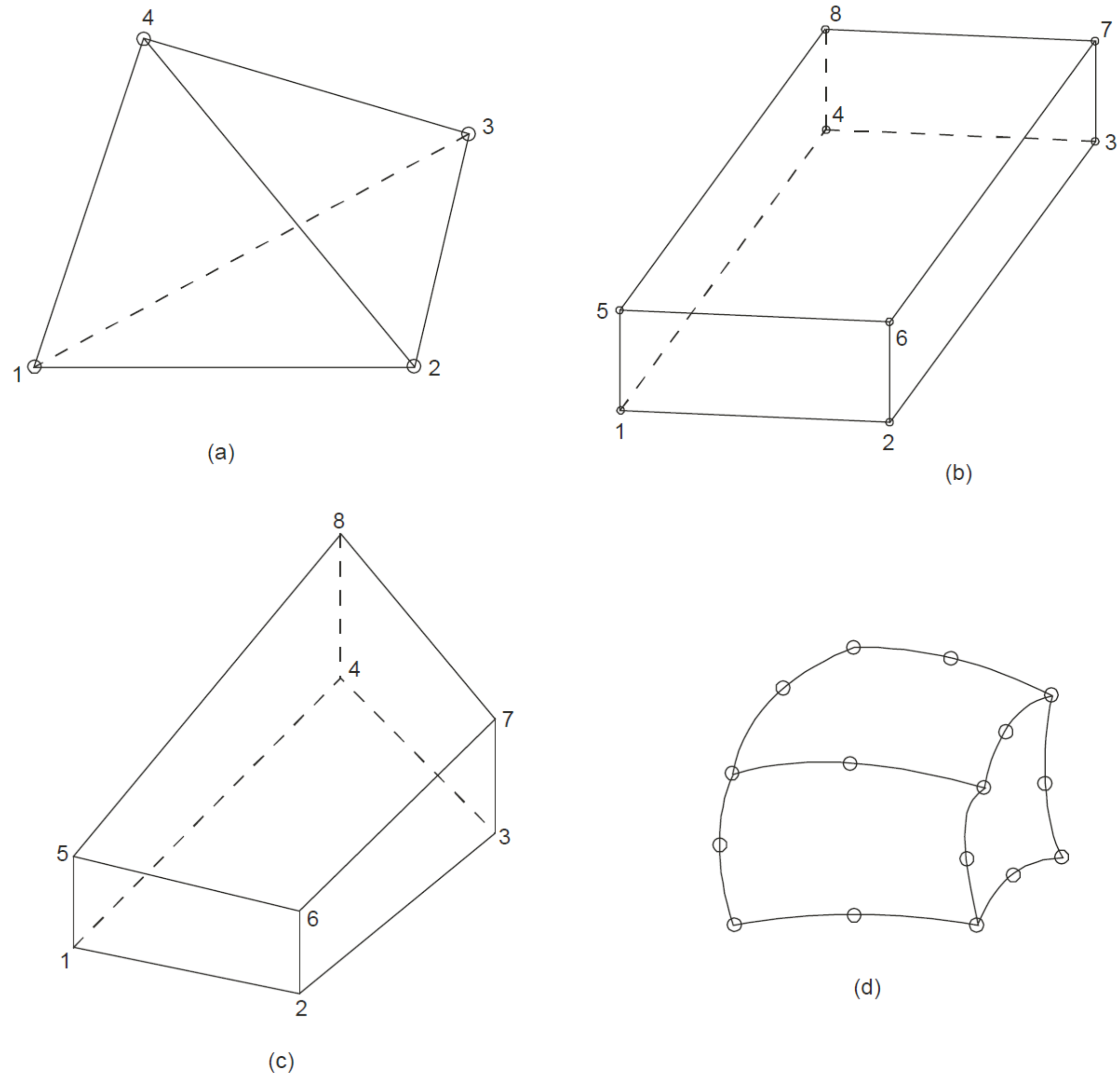
These are also known as ring type elements. These elements are useful for the analysis of axi-symmetricproblems such as analysis of cylindrical storage tanks, shafts, rocket nozzles. Axi-symmetric elements can beconstructed from one or two dimensional elements. One dimensional axi-symmetric element is a conicalfrustum and a two dimensional axi-symmetric element is a ring with a triangular or quadrilateral cross section.Two such elements are shown in figure.



Axis-symmetric elements

Three dimensional elements

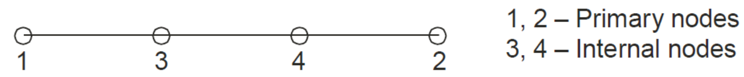
Similar to the triangle for two dimensional problems tetrahedron is the basic element for three dimensionalproblems (shown in figure). Tetrahedron is having four nodes, one at each corner. Three dimensional elements witheight nodes are either in the form of a general hexahedron or a rectangular prism, which is a particular case ofa hexahedron. The rectangular prism element is many times called as a **brick element** also. In these elementsalso one can think of using higher order elements. (shown in figure).

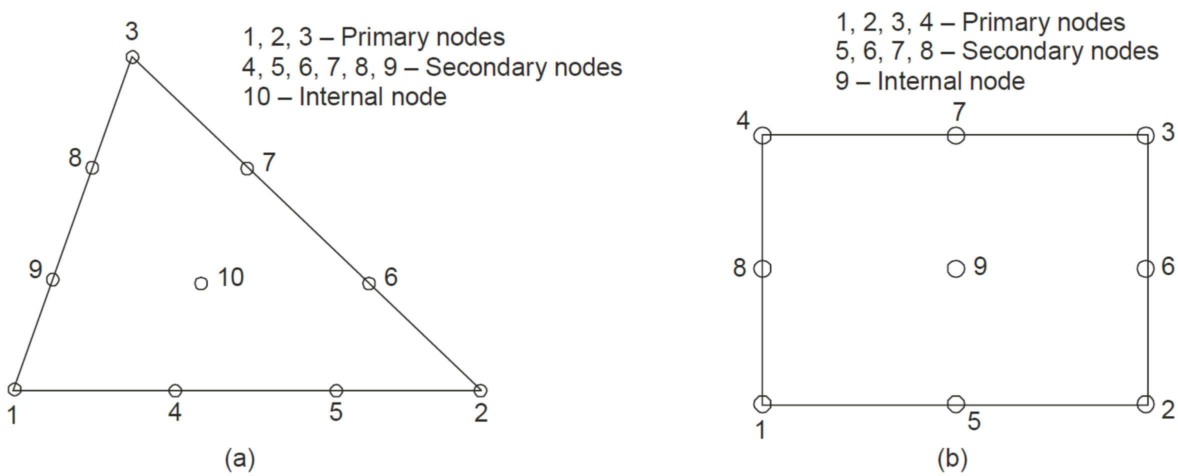


(a) Tetrahedron element (b) Rectangular prism (brick) element
(c) Arbitrary hexahedron element (d) Three dimensional quadratic element

NODES

Nodes are the selected finite points at which basic unknowns (displacements in elasticity problems) are to bedetermined in the finite element analysis. The basic unknowns at any point inside the element are determinedby using approximating/interpolation/shape functions in terms of the nodal values of the element. There aretwo types of nodes viz. external nodes and internal nodes. External nodes are those which occur on the edges/surface of the elements and they may be common to two or more elements. In figure, nodes, 1 and 2 in onedimensional element, nodes 1 to 9 in 10 noded triangular element and nodes 1 to 8 in 9 noded lagrangianelement are external nodes. These nodes may be further classified as (i) Primary nodes and (ii) Secondarynodes.





(a) 10 noded triangular element (b) 9 noded Lagrange element

Primary nodes occur at the ends of one dimensional elements or at the corners in the two or three dimensional elements. Secondary nodes occur along the side of an element but not at corners. Figure shows such nodes.

Internal nodes are the one which occur inside an element. They are specific to the element selected i.e.there will not be any other element connecting to this node. Such nodes are selected to satisfy the requirementof geometric isotropy while choosing interpolation functions. Figure shows such nodes for few typicalcases.

UNIT- V

LINEAR ELEMENT

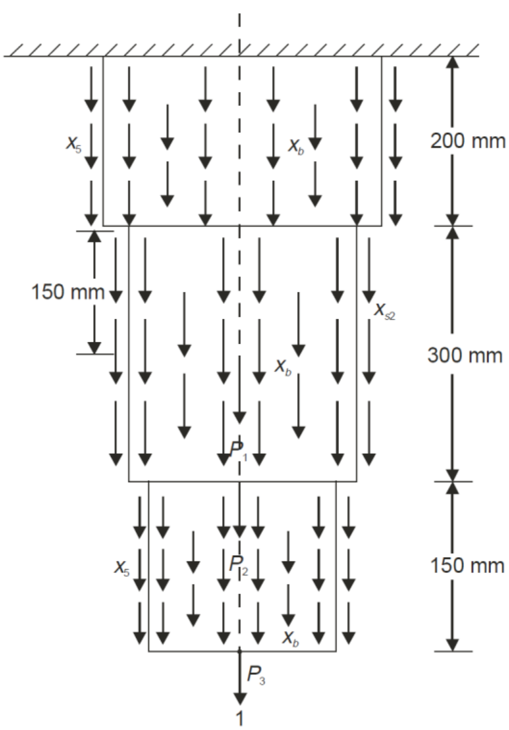
The typical member considered for explaining the procedure is shown in Fig.11.1. In this problem we seecross section varies in 3 steps A1, A2 and A3. There are three point loads P1, P2 and P3. The surface forces arexs1, xs2, and xs3 and Xb is the body force. The surface forces may be due to frictional forces, viscous drag orsurface shear. The body force is due to self weight. The material of the bar is same throughout.

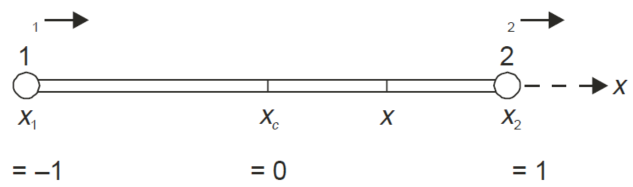
Step 1: Selecting suitable field variables and elements:

In all stress analysis problems, displacements are selected as field variables. In the tension bar or columns atany point there is only one component of displacement to be considered, i.e., the displacement in x direction.Since there is only one degree of freedom and it needs only Co continuity, we select bar element shown in figure. In this case there are only two nodes.

Step 2: Discritise the continua

In this problem there are geometric discontinuities at x = 200 mm, 500 mm and 650 mm. There is additionalpoint of discontinuity at x = 350 mm, where concentrated load P1 is acting. Hence we discritise the continua as shown in figure using four bar elements.





Hence nodal displacement vector is

$$\{\delta\} = \begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix}$$

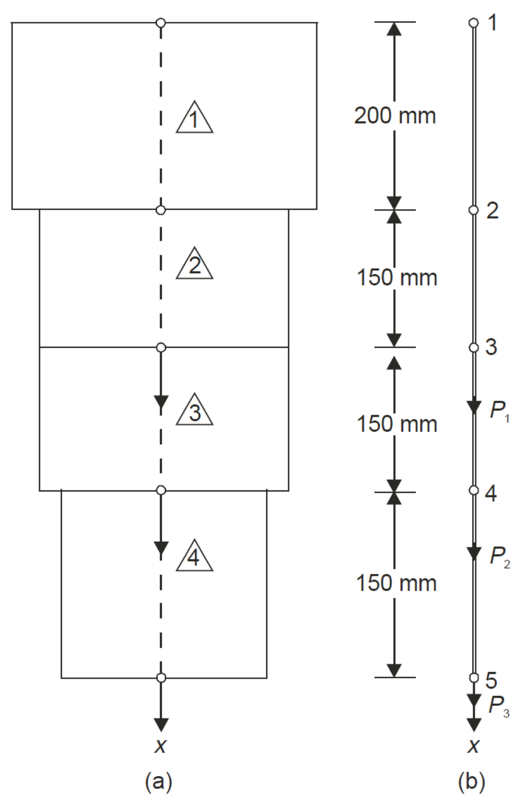
In finite element analysis the nodes may be numbered in any fashion, but to keep the band width minimum we number the nodes continuously. In this problem there are five nodes and in all such problem there is definite relationship between number of nodes and number of element i.e. Number of node = Number of elements + 1.

There is only one degree of freedom at each node. Hence total degree of freedom in the problem is

= Number of nodes × degree of freedom at each node

= 5 × 1 = 5

i.e. $$\{\delta\}^T = [\delta_1 \quad \delta_2 \quad \delta_3 \quad \delta_4 \quad \delta_5]$$



For any element local node number is 1 and 2 only, but global coordinate numbers for each element aredifferent. For example, local coordinate numbers 1 and 2 for element 3 refers to global numbering system 3and 4 respectively. The relation between the local and global node number is called connectivity details. Inthis problem the connectivity detail is as shown in figure. From this Figure it can be seen that the connectivitydetail can be easily generated also. Thus

For element (i),

Local node number 1 = i

Local node number 2 = i + 1

Element	Nodes 1	2	Local numbers
1	1	2	
2	2	3	Global numbers
3	3	4	
4	4	5	

Step 3: Select Interpolation Functions

In chapter 5 we have seen interpolation functions [N] is given by

$$\{u\} = [N]\{\delta\}_e$$

and for bar elements

$$[N] = [N_1 \; N_2], \text{ where}$$

$$N_1 = \frac{x_2 - x}{l_e} = \frac{1 - \xi}{2}$$

and

$$N_2 = \frac{x - x_1}{l_e} = \frac{1 + \xi}{2}$$

Step 4: Element Properties

In this step we assemble element stiffness matrix and nodal force vector of the element. At any point in the element,

$\{u\} = u \;\; \{\varepsilon\} = \varepsilon$ and $\{\sigma\} = \sigma,$ all in x direction, which is the only direction for these elements.

From strain displacement relations,

$$\{\varepsilon\} = \varepsilon = \frac{du}{dx} = \frac{d}{dx}\begin{bmatrix} N_1 & N_2 \end{bmatrix}\begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix} = \begin{bmatrix} \frac{dN_1}{dx} & \frac{dN_2}{dx} \end{bmatrix}\begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix}$$

$$= \frac{1}{l_e}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix}$$

$$= [B]\begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix}, \text{ where } [B] = \frac{1}{l_e}\begin{bmatrix} -1 & 1 \end{bmatrix}$$

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

$$= E\,\varepsilon, \text{ since } D = E$$

Element stiffness matrix

$$[k]_e = \iiint_v [B]^T [D][B]\, dV$$

$$= \int_0^l \frac{1}{l_e}\begin{bmatrix} -1 \\ 1 \end{bmatrix} E \frac{1}{l_e}\begin{bmatrix} -1 & 1 \end{bmatrix} A\, dx = \frac{EA}{l_e^2}\int \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} dx$$

$$= \frac{EA}{l_e^2}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}[x]_0^{l_e} = \frac{EA}{l_e}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}$$

Consistant Load

Equivalent nodal loads are to be calculated for each type of load acting on the element

(i) Body Force: X_b is the only body force in this case. In case of self weight $X_b = \rho$ where ρ is unit weight of the material. From equation 9.26 the consistant load due to this body force is given by

$$\{F\}_e = \iiint_v [N]^T \{X_b\}\, dV = \int_0^{le} \begin{Bmatrix} N_1 \\ N_2 \end{Bmatrix} \rho_b\, A\, dx$$

since $$\xi = \frac{x - x_c}{\frac{l_e}{2}} = \frac{2}{l_e}(x - x_c)$$

we get $$d\xi = \frac{2}{l_e}dx \text{ or } dx = \frac{l_e}{2}d\xi$$

and limits of integration will be from –1 to 1

$$\{F\}_e = \int_{-1}^{1} \begin{Bmatrix} \frac{1-\xi}{2} \\ \frac{1+\xi}{2} \end{Bmatrix} \rho_b \ A \frac{l_e}{2} \, d\xi$$

Now $\int \frac{1-\xi}{2} \rho_b \, A \frac{l_e}{2} d\xi = \frac{l_e}{4} AX_b \left[\xi - \frac{\xi^2}{2} \right]_{-1}^{1} = \frac{l_e}{2} A \rho_b$

Similarly $\int \frac{1+\xi}{2} \rho_b \, A \frac{l_e}{2} d\xi = \frac{1}{2} A l_e \, \rho_b$

$$\therefore \{F\}_e = \frac{A l_e \, \rho_b}{2} \begin{bmatrix} 1 \\ 1 \end{bmatrix}$$

Noting that Al_e is volume of the element, we find that half the self weight goes to each node.

(ii) Surface Load: If X_s is the intensity of surface load, $T = X_s \times$ perimeter is the load per unit length of the element. Then consistant load corresponding to it is

$$\{F\}_e = \iint \{N\}^T X_s \, ds$$

$$= \int_0^l \begin{Bmatrix} N_1 \\ N_2 \end{Bmatrix} T \, dx = \int_{-1}^{l} \begin{Bmatrix} N_1 \\ N_2 \end{Bmatrix} T_s \frac{l_e}{2} d\xi$$

$$= \int_{-1}^{l} \begin{Bmatrix} \frac{1-\xi}{2} \\ \frac{1+\xi}{2} \end{Bmatrix} T_s \frac{l_e}{2} d\xi = \frac{T l_e}{2} \begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

Thus the consistant load for such surface traction is also half the total load at each node.

(iii) Point Load: Point loads can be directly added to nodal force vector.

After finding consistant load due to all types of loads, element nodal force vector $\{F\}_e = \begin{Bmatrix} F_{e1} \\ F_{e2} \end{Bmatrix}$ can be assembled.

Step 5: Global Properties

From step 3, we have

$$[k]_{e_1} = \frac{EA_1}{l_{e1}} \overset{\begin{matrix} 1 & \quad 2 \end{matrix}}{\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}} \begin{matrix} 1 \\ 2 \end{matrix} \qquad [k]_{e_2} = \frac{EA_2}{l_{e2}} \overset{\begin{matrix} 2 & \quad 3 \end{matrix}}{\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}} \begin{matrix} 2 \\ 3 \end{matrix}$$

$$[k]_{e_3} = \frac{EA_3}{l_{e3}} \overset{\begin{matrix} 3 & \quad 4 \end{matrix}}{\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}} \begin{matrix} 3 \\ 4 \end{matrix} \qquad [k]_{e_4} = \frac{EA_4}{l_{e4}} \overset{\begin{matrix} 4 & \quad 5 \end{matrix}}{\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}} \begin{matrix} 4 \\ 5 \end{matrix}$$

For each element their position corresponding to global rows and columns are indicated above. Now global stiffness matrix $\{k\}$ of size 5×5 is to be assembled. First this is made a null matrix and then one by element stiffness matrix is added to corresponding element in global matrix. After first element stiffness matrix is placed in global stiffness matrix, it looks as-

$$E\begin{bmatrix} \frac{A_1}{l_{e1}} & -\frac{A_1}{l_{e1}} & 0 & 0 & 0 \\ -\frac{A_1}{l_{e1}} & \frac{A_1}{l_{e1}} & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

After second element stiffness is placed in global stiffness matrix, it looks as

$$E\begin{bmatrix} \frac{A_1}{l_{e1}} & -\frac{A_1}{l_{e1}} & 0 & 0 & 0 \\ -\frac{A_1}{l_{e1}} & \frac{A_1}{l_{e1}} + \frac{A_2}{l_{e1}} & -\frac{A_2}{l_{e2}} & 0 & 0 \\ 0 & -\frac{A_2}{l_{e2}} & \frac{A_2}{l_{e2}} & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix}$$

Final stiffness matrix in global system

$$K = E\begin{bmatrix} \frac{A_1}{l_{e1}} & -\frac{A_1}{l_{e1}} & 0 & 0 & 0 \\ -\frac{A_1}{l_{e1}} & \frac{A_1}{l_{e1}} + \frac{A_2}{l_{e2}} & -\frac{A_2}{l_{e2}} & 0 & 0 \\ 0 & -\frac{A_2}{l_{e2}} & \frac{A_2}{l_{e2}} + \frac{A_3}{l_{e3}} & -\frac{A_3}{l_{e3}} & 0 \\ 0 & 0 & -\frac{A_3}{l_{e3}} & \frac{A_3}{l_{e3}} + \frac{A_4}{l_{e4}} & -\frac{A_4}{l_{e4}} \\ 0 & 0 & 0 & -\frac{A_4}{l_{e4}} & \frac{A_4}{l_{e4}} \end{bmatrix}$$

Thus we find the stiffness matrix is a symmetric matrix and its half the band width is equal to maximum difference in nodes of any element multiplied by degrees of freedom at each node plus 1, that is 2 in this problem

Load Vector {F}

Load vector $\{F\}^T = [F_1 \quad F_2 \quad F_3 \quad F_4 \quad F_5]$

Let the element load vectors be

$$\{F\}_{e1} = \begin{Bmatrix} F_{11} \\ F_{12} \end{Bmatrix}; \quad \{F\}_{e2} = \begin{Bmatrix} F_{21} \\ F_{22} \end{Bmatrix}$$

$$\{F\}_{e3} = \begin{Bmatrix} F_{31} \\ F_{32} \end{Bmatrix}; \quad \{F\}_{e4} = \begin{Bmatrix} F_{41} \\ F_{42} \end{Bmatrix}$$

Then global load vector $\{F\}$ is given by

$$\{F\} = \begin{Bmatrix} F_{11} \\ F_{12} + F_{21} \\ F_{22} + F_{31} \\ F_{32} + F_{41} \\ F_{42} \end{Bmatrix}$$

Thus we can assemble global / structure stiffness equation as

$$\underset{5x5}{[k]} \; \underset{5x1}{\{\delta\}} = \underset{5x1}{\{F\}}$$

Step 6: Boundary Conditions

In this problem there is only one boundary condition i.e. $\delta_1 = 0$ or it may have specified value. There are two methods of imposing the boundary conditions:

(i) Elimination Approach

(ii) Penalty Approach

Step 7: Solution of Simultaneous Equations

After imposing the boundary conditions, the simultaneous equations 11.13 are to be solved. Any method of solving simultaneous equations can be employed. Gauss elimination is commonly employed. In many programs to save the memory in storing stiffness matrix *k*, half the band width of the matrix is stored and Choleski's decomposition method employed. The solution gives the unknown nodal values.

Step 8: Additional Calculations

The additional calculations required may be to find strains and stresses at various points. These calculations are carried out element by element. From the list of global nodal values δ, for each element nodal values δ_1 and δ_2 of the element under consideration is picked up. Then displacement within the element.

$$u = [N]\{\delta\}_e = [N_1 \;\; N_1]\begin{Bmatrix} \delta_1 \\ \delta_2 \end{Bmatrix}$$

since 'ξ' coordinate of the point under consideration is known 'u' can be found. Then

$$\{\varepsilon\} = \varepsilon = [B]\{\delta\}_e$$

and

$$\{\sigma\} = \sigma = [D]\{\varepsilon\}_e = E\varepsilon$$

$$= E[B]\{\delta\}_e$$

Calculation of Reactions

Another important stress resultant required in the stress analysis is the reactions at support. This can be found from the equilibrium conditions of the support. For example, in this problem support is at node 1 and at this point displacement δ_1 is zero. Hence if R_1 is the reaction of the support in direction 1,then

$$k_{11}\delta_1 + k_{12}\delta_2 + k_{13}\delta_3 + k_{14}\delta_4 + k_{15}\delta_5 = F_1 + R_1$$

or

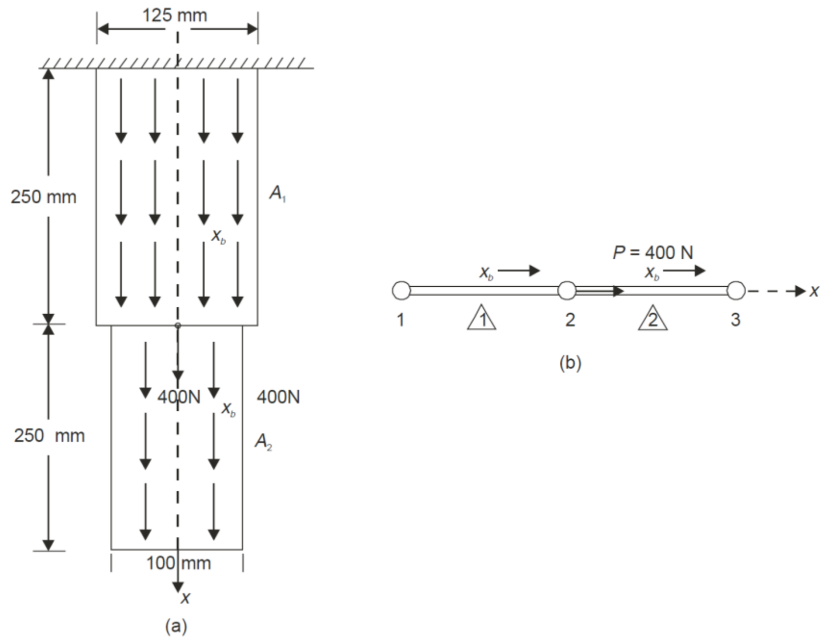
$$R_1 = k_{11}\delta_1 + k_{12}\delta_2 + k_{13}\delta_3 + k_{14}\delta_4 + k_{15}\delta_5 - F_1$$

In general $R_1 = k_{11}\delta_1 + k_{12}\delta_2 + \ldots + k_{1N}\delta_N - F_1$

Where N is total number of nodal displacements

Example:

The thin plate of uniform thickness 20 mm, is as shown in Fig. 11.5(a). In addition to the self weight, the plate is subjected to a point load of 400N at mid-depth. The Young's modulus $E = 2 \times 10^5$ N/mm^2 and unit weight $\rho = 0.8 \times 10^{-4}$ N/mm^2. Analyse the plate after modeling it with two elements and find the stresses in each element. Determine the support reactions also.



Solution:

$A_1 = 125 \times 20 = 2500$ mm²

$A_2 = 100 \times 20 = 2000$ mm²

The plate is modeled with two elements as shown in Fig. 11.5 (b)

$$[k]_{e1} = \frac{EA_1}{l_{e1}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{2 \times 10^5 \times 2500}{250}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 2 \times 10^5 \begin{bmatrix} 10 & -10 \\ -10 & 10 \end{bmatrix}$$

$$[k]_{e2} = \frac{EA_2}{l_{e2}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = \frac{2 \times 10^5 \times 2000}{250}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 2 \times 10^5 \begin{bmatrix} 8 & -8 \\ -8 & 8 \end{bmatrix}$$

$$\therefore\ [k] = \begin{bmatrix} 10 & -10 & 0 \\ -10 & 10+8 & -8 \\ & -8 & 8 \end{bmatrix} = \begin{bmatrix} 10 & -10 & 0 \\ -10 & 18 & -8 \\ 0 & -8 & 8 \end{bmatrix}$$

Consistant Loads: Due to body force only

$$\{F\}_e = \begin{Bmatrix} F_{e1} \\ F_{e2} \end{Bmatrix} = X_b \frac{Al_e}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix}$$

$$\{F\}_{e1} = \begin{Bmatrix} F_{11} \\ F_{12} \end{Bmatrix} = \frac{0.8 \times 10^{-4} \times 2500 \times 250}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} = \begin{Bmatrix} 25 \\ 25 \end{Bmatrix}$$

$$\{F\}_{e2} = \begin{Bmatrix} F_{21} \\ F_{22} \end{Bmatrix} = \frac{0.8 \times 10^{-4} \times 2000 \times 250}{2}\begin{Bmatrix} 1 \\ 1 \end{Bmatrix} = \begin{Bmatrix} 20 \\ 20 \end{Bmatrix}$$

Apart form these there is a 400N concentrated load at node 2. Hence,

$$\{F\} = \begin{bmatrix} 25 \\ 25 + 20 + 400 \\ 20 \end{bmatrix} = \begin{Bmatrix} 25 \\ 445 \\ 20 \end{Bmatrix}$$

Hence the stiffness equation is,

$$2 \times 10^5 \begin{bmatrix} 10 & -10 & 0 \\ -10 & 18 & -8 \\ 0 & -8 & 8 \end{bmatrix} \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{Bmatrix} = \begin{Bmatrix} 25 \\ 445 \\ 20 \end{Bmatrix}$$

The boundary condition is $\delta_1 = 0$. Hence the reduced equation is,

$$2 \times 10^5 \begin{bmatrix} 18 & -8 \\ -8 & 8 \end{bmatrix} \begin{Bmatrix} \delta_2 \\ \delta_3 \end{Bmatrix} = \begin{bmatrix} 445 - 10 \times 0 \\ 20 - 0 \times 0 \end{bmatrix} = \begin{Bmatrix} 445 \\ 20 \end{Bmatrix}$$

$$2 \times 10^5 \begin{bmatrix} 18 & -8 \\ 0 & 8 - \frac{8}{18} \times 8 \end{bmatrix} \begin{Bmatrix} \delta_2 \\ \delta_3 \end{Bmatrix} = \begin{bmatrix} 445 \\ 20 + \frac{8}{18} \times 445 \end{bmatrix}$$

i.e.

$$2 \times 10^5 \begin{bmatrix} 18 & -8 \\ 0 & 4.444 \end{bmatrix} \begin{Bmatrix} \delta_2 \\ \delta_3 \end{Bmatrix} = \begin{bmatrix} 445 \\ 217.778 \end{bmatrix}$$

$$\therefore\ \delta_3 = \frac{217.778}{4.444 \times 2 \times 10^5} = 2.45 \times 10^{-4} \text{ mm}$$

from equation 1, we have

$$2 \times 10^5 \left[18\delta_2 - 8\delta_3 \right] = 445$$

$$2 \times 10^5 \left[18\delta_2 - 8 \times 2.45 \times 10^{-4} \right] = 445$$

$$18\delta_2 - 1.96 \times 10^{-3} = 2.225 \times 10^{-3}$$

$$\delta_2 = 2.325 \times 10^{-4} \text{ mm}$$

from the relation

$$\sigma = E[B]\{\delta\}_e \text{ we get,}$$

$$\sigma_1 = 2 \times 10^5 \frac{1}{250}[-1 \quad 1]\begin{Bmatrix} 0 \\ 2.325 \times 10^{-4} \end{Bmatrix} = 0.186 \text{ N/mm}^2$$

$$\sigma_2 = 2 \times 10^5 \frac{1}{250}[-1 \quad 1]\begin{Bmatrix} 2.325 \times 10^{-4} \\ 2.45 \times 10^{-4} \end{Bmatrix} = 0.01 \text{ N/mm}^2$$

Reaction at Support:

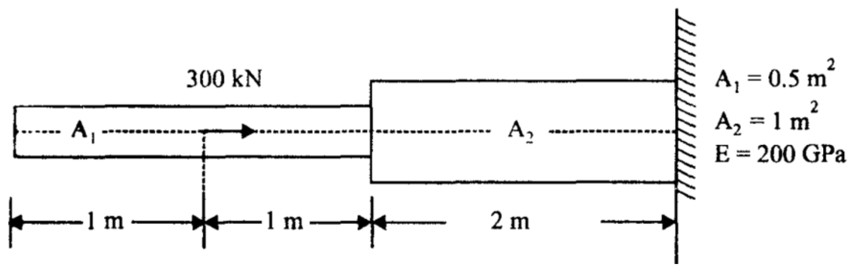
$$R_1 = [k_{11} \quad k_{12} \quad k_{13}]\begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{Bmatrix} - F_1 = 2 \times 10^5 [10 \quad -10 \quad 0]\begin{Bmatrix} 0 \\ 2.325 \times 10^{-4} \\ 2.45 \times 10^{-4} \end{Bmatrix} - 25$$

$$\therefore \; R_1 = 490\text{N}$$

[Obviously in this simple problem reaction = total load].

Example:

Determine the nodal displacements and element stresses by finite element formulation for the following figure. Use P=300 k N; A_1=0.5 m^2; A_2=1 m^2; E=200 GPa



Solution

The structure is modeled with 3 axial loaded elements connected by nodes 1-2, 2-3 and 3-4 as shown below

1 ——————— 2 —————— 3 ——————— 4

Stiffness matrices of elements 1, 2 and 3 are given by

$$[K_1]=\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} [K_2]=\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} [K_3]=\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}$$

where, $k_1 = A_1E/L_1 = 0.5 \times 200 \times 10^9/1.0 = 1.0 \times 10^{11}$

and $k_2 = A_2\,E/L_2 = 1 \times 200 \times 10^9/2.0 = 1.0 \times 10^{11}$

Assembled stiffness matrix is obtained by adding corresponding terms as,

$$k_1 = [K] = \begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & k_1+k_2 & -k_1 & 0 \\ 0 & -k_1 & k_1+k_2 & -k_2 \\ 0 & 0 & -k_2 & k_2 \end{bmatrix} = 1.0\times10^{11}\begin{bmatrix} 1 & -1 & 0 & 0 \\ -1 & 2 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix}$$

Corresponding assembled nodal load vector and nodal displacement vector are

$$P=\begin{Bmatrix} 0 \\ 300{,}000 \\ 0 \\ R \end{Bmatrix}; q=\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix}$$

After applying boundary condition, $u_4=0$, the fourth row and fourth column are removed resulting in

$$1.0\times10^{11}\begin{bmatrix}1 & -1 & 0\\ -1 & 2 & -1\\ 0 & -1 & 2\end{bmatrix}\begin{Bmatrix}u_1\\ u_2\\ u_3\end{Bmatrix}=\begin{Bmatrix}0\\ 300,000\\ 0\end{Bmatrix}$$

Solving the above set of equations gives,

$u_1 = 6 \times 10^{-6}$ m; $u_2 = 6 \times 10^{-6}$ m; $u_3 = 3 \times 10^{-6}$ m

Stress in element-1,

$$\sigma_1 = E\,[B_1]\,\{q_1\} = E\,[-1/L_1 \quad 1/L_1]\begin{Bmatrix}u_1\\ u_2\end{Bmatrix} = 0\ \text{N/m}^2$$

Stress in element-2,

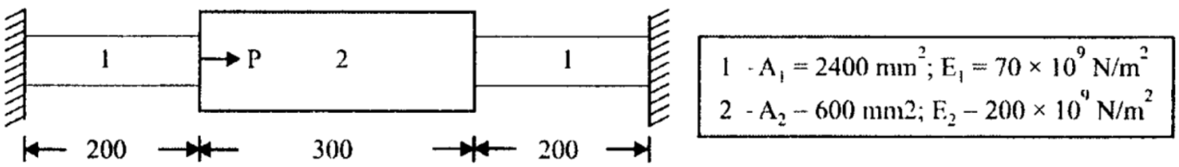
$$\sigma_2 = E\,[B_2]\,\{q_2\} = E\,[-1/L_1 \quad 1/L_1]\begin{Bmatrix}u_2\\ u_3\end{Bmatrix} = -6 \times 10^5\ \text{N/m}^2$$

Stress in element-3,

$$\sigma_3 = E\,[B_3]\,\{q_3\} = E\,[-1/L_2 \quad 1/L_2]\begin{Bmatrix}u_3\\ u_4\end{Bmatrix} = -3 \times 10^5\ \text{N/m}^2$$

Example:

An axial load $P = 200 \times 10^3$ N is applied on a bar as shown. Using the penalty approach for handling boundary conditions, determine nodal displacements, stress in each material and reaction forces.



Solution

1 ———— 2 ———— 3 ———— 4

Considering a 3-element truss model, stiffness matrices of elements 1, 2 and 3 (connected by nodes 1, 2; 2, 3 and 3, 4 respectively) are given by,

$$[K_1] = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix}; \quad [K_3] = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix}; \quad [K_2] = \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix}$$

where

$$k_1 = A_1 E_1 / L_1 = 2400 \times 70 \times 10^3/200 = 84 \times 10^4$$

and $k_2 = A_2 E_2 / L_2 = 600 \times 200 \times 10^3/300 = 40 \times 10^4$

Assembled stiffness matrix is obtained by adding corresponding terms as,

$$[K] = \begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & k_1 + k_1 & -k_1 & 0 \\ 0 & -k_1 & k_1 + k_2 & -k_2 \\ 0 & 0 & -k_2 & k_2 \end{bmatrix} = 10^4 \begin{bmatrix} 84 & -84 & 0 & 0 \\ -84 & 84+40 & -40 & 0 \\ 0 & -40 & 40+80 & -84 \\ 0 & 0 & -84 & 84 \end{bmatrix}$$

Corresponding assembled nodal load vector and nodal displacement vector are

$$P = \begin{Bmatrix} 0 \\ 200{,}000 \\ 0 \\ 0 \end{Bmatrix}; \qquad q = \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix}$$

For the penalty approach, $C = \max(k_{ij}) \times 10^4 = 124 \times 10^4$

Since the bar is fixed at nodes 1 and 4, the equations are then modified using C as,

$$\begin{Bmatrix} 0 \\ 200{,}000 \\ 0 \\ 0 \end{Bmatrix} = 10^4 \begin{bmatrix} 84+124\times10^4 & -84 & 0 & 0 \\ -84 & 124 & -40 & 0 \\ 0 & -40 & 124 & -84 \\ 0 & 0 & -84 & 84+124\times10^4 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix}$$

From 4th eqn. $0 = 10^4 [-84\, u_3 + (84 + 124 \times 10^4)\, u_4]$

or $u_4 = 6.7737 \times 10^{-5}\, u_3$

From 3rd eqn $0 = 10^4 [-40\, u_2 + 124\, u_3 - 84\, u_4]$

substituting for u_4 from the above,

$$u_3 = 0.3226\, u_2$$

2^{nd} eqn now becomes,

$$200{,}000 = 10^4\ [-84\ u_1 + 124\ u_2 - 40\ u_3]$$

or $$-0.64u_1 + 1.111u_2 = 0.2$$

1^{st} equation gives,

$$0 = 10^4\ [(84 + 124 \times 10^4)\ u_1 \quad -84\ u_2\ ]$$

From these two equations,

$$u_1 = 1.2195 \times 10^{-5}\ \text{mm}; \quad u_2 = 0.180034\ \text{mm}$$

Substituting in 3^{rd} and 4^{th} eqn.,

$$u_3 = 0.058079\ \text{mm}; \qquad u_4 = 3.9341 \times 10^{-6}\ \text{mm}$$

Reactions, $$R_1 = -Cu_1 = (124 \times 10^8).(1.2195 \times 10^{-5}) = -151.22 \times 10^3\ \text{N}$$

$$R_2 = -Cu_4 = (124 \times 10^8).(3.9341 \times 10^{-6}) = -48.78 \times 10^3\ \text{N}$$

Stresses in the elements,

$$\sigma_1 = E_1\varepsilon_1 = E_1B_1q_{1\text{-}2}$$

$$= 70 \times 10^3 \begin{bmatrix} \frac{-1}{200} & \frac{1}{200} \end{bmatrix} \begin{Bmatrix} 1.2195 \times 10^{-5} \\ 0.180034 \end{Bmatrix}$$

$$= 63.01\ \text{N/mm}^2$$

$$\sigma_2 = E_2\varepsilon_2 = E_2B_2q_{2\text{-}3}$$

$$= 200 \times 10^3 \begin{bmatrix} \frac{-1}{300} & \frac{1}{300} \end{bmatrix} \begin{Bmatrix} 0.180034 \\ 0.058079 \end{Bmatrix}$$

$$= -81.3\ \text{N/mm}^2$$

$$\sigma_3 = E_3\varepsilon_3 = E_1B_1q_{3\text{-}4}$$

$$= 70 \times 10^3 \begin{bmatrix} \frac{-1}{200} & \frac{1}{200} \end{bmatrix} \begin{Bmatrix} 0.058079 \\ 3.9341 \times 10^{-6} \end{Bmatrix}$$

$$= -20.3\ \text{N/mm}^2$$

Elimination method

Since the bar is fixed at nodes 1 and 4, corresponding rows and columns of the assembled stiffness matrix are deleted, resulting in $\{P\}_R = [K]_R \{u\}_R$

or $$\begin{Bmatrix} 200,000 \\ 0 \end{Bmatrix} = 10^4 \begin{bmatrix} 124 & -40 \\ -40 & 124 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}$$

Solving these two simultaneous equations, we get

$$u_2 = 155 / 861 = 0.180023 \text{ mm}$$

and $$u_3 = 50 / 861 = 0.058072 \text{ mm}$$

Reactions can now be obtained by substituting the nodal displacements in the deleted equations of the assembled stiffness matrix.

$$R_1 = 10^4 \,[(84 + 124 \times 10^4) \quad -84 \quad 0 \quad 0] \;[u_1 \quad u_2 \quad u_3 \quad u_4]^T$$

$$= -84 \times 10^4 \, u_2 \qquad = -84 \times 10^4 \,(155/861) \qquad = 151219 \text{ N}$$

$$R_4 = 10^4 \,[\,0 \quad 0 \quad -84 \quad (84 + 124 \times 10^4)\,]\,[\,u_1 \quad u_2 \quad u_3 \quad u_4]^T$$

$$= -84 \times 10^4 \, u_3 \qquad = -84 \times 10^4 \,(50/861) \qquad = 48780 \text{ N}$$

These reaction values are identical to hose obtained by the penalty approach

Check : For force equilihrium of the structure,

$$R_1 + R_4 = \text{Applied load } P \approx 200 \text{ kN}$$

This equation is satisfied with the results obtained

Note that results by penalty approach match very closely with those by elimination approach.

Example:

Consider the truss element with the coordinates 1 (10,10) and 2 (50,40). If the displacement vector is $q = [15 \ \ 10 \ \ 21 \ \ 43]^T$ mm, then determine (i) the vector q' (ii) stress in the element and (iii) stiffness matrix if E=70 GPa and A=200 mm^2

Solution :

(i) The nodal displacement vector in local coordinate system

$$\{q'\} = \begin{bmatrix} l & m & 0 & 0 \\ 0 & 0 & l & m \end{bmatrix} \{q\}$$

where $l = (x_2 - x_1)/L$ and $m = (y_2 - y_1)/L$ are the direction cosines of the element

Length of the element,

$$L = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2} = \sqrt{(50-10)^2 + (40-10)^2} = 50 \text{ mm}$$

$$l = \left(\frac{50-10}{50}\right) = \frac{4}{5}; \qquad m = \frac{(40-10)}{50} = \frac{3}{5}$$

$$\{q'\} = \begin{bmatrix} 4/5 & 3/5 & 0 & 0 \\ 0 & 0 & 4/5 & 3/5 \end{bmatrix} \begin{Bmatrix} 15 \\ 10 \\ 21 \\ 43 \end{Bmatrix} = \begin{Bmatrix} 90/5 \\ 213/5 \end{Bmatrix}$$

(ii) Stress in the element, $\sigma = E\,\varepsilon = E\begin{bmatrix} \frac{-1}{L} & \frac{1}{L} \end{bmatrix}\{q'\}$

$$= 70{,}000 \begin{bmatrix} \frac{-1}{50} & \frac{1}{50} \end{bmatrix} \begin{Bmatrix} 90/5 \\ 213/5 \end{Bmatrix}$$

$$= 34.44 \times 10^3 \text{ N/mm}^2$$

(iii) Stiffness matrix of the element,

$$[K]=\frac{AE}{L}\begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix}=\frac{200\times 70{,}000}{50\times 25}\begin{bmatrix} 16 & 12 & -16 & -12 \\ 12 & 9 & -12 & -9 \\ -16 & -12 & 16 & 12 \\ -12 & -9 & 12 & 9 \end{bmatrix}$$

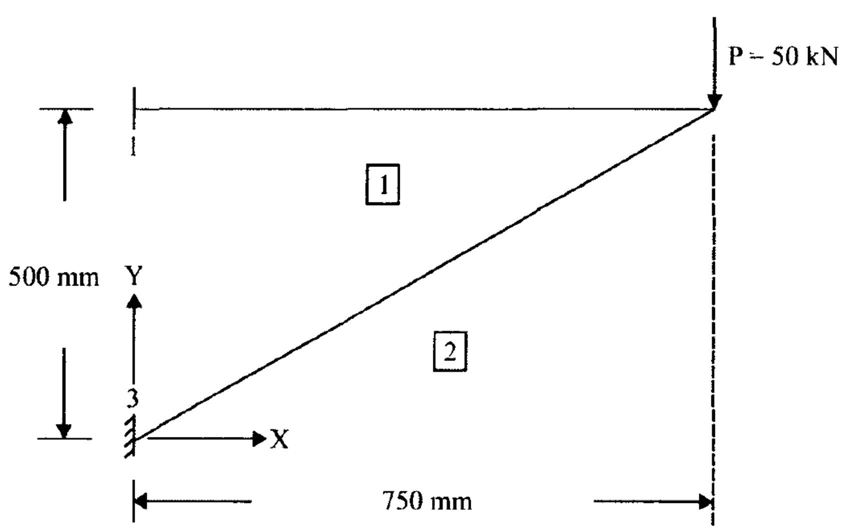
Example:

Determine the stiffness matrix, stresses and reactions in the truss structure shown below, assuming points 1 and 3 are fixed. Use $E = 200$ GPa and $A = 1000\ mm^2$.

Solution

Stiffness matrix of any truss element is given by

$$[K]=\frac{AE}{L}\begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix}$$



In the given problem, $L_1 = 750$ mm; $L_2 = \sqrt{[750^2 + 500^2]} = 250\sqrt{13}$

For element-1, $l = \frac{(x_2 - x_1)}{L_1} = 1$ and $m = \frac{y_2 - y_1}{L_1} = 0$

$$[K]_1 = \frac{AE}{750}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$\frac{AE}{750} = 266.67 \times 10^3$$

For element-2, $l = \frac{(x_3 - x_2)}{L_2} = \frac{3}{\sqrt{13}}$ and $m = \frac{y_3 - y_2}{L_2} = \frac{2}{\sqrt{13}}$

$$[K]_2 = \frac{AE}{250 \times 13 \times \sqrt{13}}\begin{bmatrix} 9 & 6 & -9 & -6 \\ 6 & 4 & -6 & -4 \\ -9 & -6 & 9 & 6 \\ -6 & -4 & 6 & 4 \end{bmatrix}$$

$$\frac{AE}{250 \times 13\sqrt{13}} = 17.07 \times 10^3$$

The assembled stiffness matrix is given by appropriate addition of stiffness coefficients of the two elements,

$$[K]=10^3\begin{bmatrix} 266.67 & 0 & -266.67 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -266.67 & 0 & 266.67-153.63 & 102.42 & -153.63 & -102.42 \\ 0 & 0 & -102.42 & 68.28 & -102.42 & -68.28 \\ 0 & 0 & -153.63 & -102.42 & 153.63 & 102.42 \\ 0 & 0 & -102.42 & -68.28 & 102.42 & 68.28 \end{bmatrix}$$

After applying boundary conditions that $u_1 = v_1 = u_3 = v_3 = 0$, the load-displacement relationships reduce to $\{P\}_R = [K]_R \{u\}_R$

$$\begin{Bmatrix} 0 \\ -50000 \end{Bmatrix} = 10^3 \begin{bmatrix} -266.67+153.63 & 102.42 \\ 102.42 & 68.28 \end{bmatrix} \begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix}$$

Solving these two simultaneous equations gives

$u_2 = 0.2813$ mm and $v_2 = -1.154$ mm

Displacements of element-1 in local coordinate system are given by

$$\{q_1'\} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0.2813 \\ -1.154 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0.2813 \end{Bmatrix}$$

Stress in element-1, $\sigma_1 = E\,\varepsilon_1 = E[-1/L \quad 1/L]\{q_1\}$

$$= 200 \times 10^3 \times 0.2813 / 750 = 75 \text{ N/mm}^2$$

Displacements of element-2 in local coordinate system are given by

$$\{q_2'\}=\begin{bmatrix}3/\sqrt{13} & 2/\sqrt{13} & 0 & 0\\ 0 & 0 & 3\sqrt{13} & 2\sqrt{13}\end{bmatrix}\begin{Bmatrix}0.28313\\ -1.154\\ 0\\ 0\end{Bmatrix}=\begin{Bmatrix}-0.406\\ 0\end{Bmatrix}$$

Stress in element-2, $\sigma_2 = E\,\varepsilon_2 = E\begin{bmatrix}\dfrac{-1}{L} & \dfrac{1}{L}\end{bmatrix}\{q_2'\}$

$$=200\times10^3\times\frac{(-0.406)}{250\sqrt{13}}=90.08\ \text{N/mm}^2$$

Reactions at the two fixed ends are obtained from the equations of the ssembled stiffness matrix corresponding to the specified zero displacements

$$\begin{Bmatrix}R_{1-X}\\ R_{1-Y}\\ R_{1-X}\\ R_{1-Y}\end{Bmatrix}=10^3\begin{bmatrix}266.67 & 0 & -266.67 & 0 & 0 & 0\\ 0 & 0 & 0 & 0 & 0 & 0\\ 0 & 0 & -153.63 & -102.42 & 153.63 & 102.42\\ 0 & 0 & -102.42 & -68.28 & 102.42 & 68.28\end{bmatrix}\begin{Bmatrix}0\\ 0\\ 0.2813\\ -1.154\\ 0\\ 0\end{Bmatrix}$$

$$=\begin{Bmatrix}-75014.3\\ 0\\ 74976.6\\ 49984.4\end{Bmatrix}\text{N}$$

The exact solution can be obtained from the equilibrium conditions as follows -

The force in element-2 is such that its vertical component is equal to the applied load P. Horizontal component of this force is given by

P x (750/500) = 75000 N

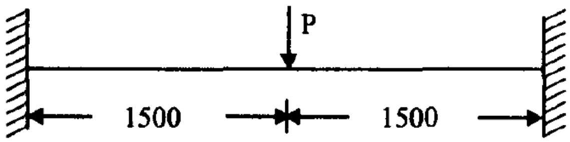
$R_{3-Y} + P = 0$ or $R_{3-Y} = 50{,}000$ N

$R_{3-X} + R_{1-X} = 0$ or $R_{1-X} = -R_{3-X} = 75{,}000$ N

It can be seen that the approximate solution obtained by FEM is in close agreement with the exact solution obtained from equilibrium consideration.

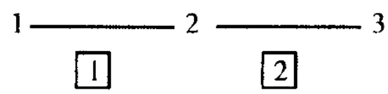
Example:

A concentrated load P = 50 kN is applied at the center of a fixed beam of length 3m, depth 200 mm and width 120 mm. Calculate the deflection and slope at the mid point. Assume $E = 2 \times 10^5$ N/mm^2.



Solution

The finite element model consists of 2 beam elements, as shown here, with nodes 1 and 3 at the two fixed supports and node 2 at the location where load P is applied.



Stiffness matrices of elements 1 and 2 (connected by nodes 1 and 2 ; 2 and 3 respectively, each with L = 1500 mm) are given by,

$$[K]=\frac{E\,I_z}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} = \frac{2\times10^5\times\dfrac{(120\times200^3)}{12}}{L^3}\begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}$$

Assembling the element stiffness matrices, we get

$$\begin{Bmatrix} P_1 \\ M_1 \\ P_2 \\ M_2 \\ P_3 \\ M_3 \end{Bmatrix} = \frac{2\times10^5\times\dfrac{120\times200^3}{12}}{1500^3}\begin{bmatrix} 12 & 6L & -12 & 6L & 0 & 0 \\ 6L & 4L^2 & -6L & 2L^2 & 0 & 0 \\ -12 & -6L & 12+12 & -6L+6L & -12 & 6L \\ 6L & 2L^2 & -6L+6L & 4L^2+4L^2 & -6L & 2L^2 \\ 0 & 0 & -12 & -6L & 12 & -16L \\ 0 & 0 & 6L & 2L^2 & -6L & 4L^2 \end{bmatrix}\begin{Bmatrix} w_1 \\ \theta_1 \\ w_2 \\ \theta_2 \\ w_3 \\ \theta_3 \end{Bmatrix}$$

After applying boundary conditions $v_1 = v_3 = 0$ and $(\theta_z)_1 = (\theta_z)_3 = 0$, the equations reduce to

$$\begin{Bmatrix} P_2 \\ M_2 \end{Bmatrix} = \frac{2\times10^5 \times \frac{(120\times200^3)}{12}}{1500^3} \begin{bmatrix} 12+12 & -6L+6L \\ -6L+6L & 4L^2+4L^2 \end{bmatrix} \begin{Bmatrix} v_2 \\ (\theta_z)_2 \end{Bmatrix}$$

The applied loads are $P_2 = -50000$ N and $M_2 = 0$

Therefore, $v_2 = \dfrac{-50000\times1500^3}{\left[2\times10^5 \times \frac{(120\times200^3)}{12} \times 24\right]} = -0.4395$ mm

and $(\theta_z)_2 = 0$

Check : From strength of materials approach, $v_3 = \dfrac{-PL^3}{24\,EI}$ or $\dfrac{P(2L)^3}{192\,EI}$

$= -0.4395$ mm

and the deflection being symmetric, slope at the center $(\theta_z)_2 = 0$.

Example:

Consider the bar shown in Fig. E3.4. An axial load $P = 200 \times 10^3$ N is applied as shown. Using the penalty approach for handling boundary conditions, do the following:

(a) Determine the nodal displacements.
(b) Determine the stress in each material.
(c) Determine the reaction forces.

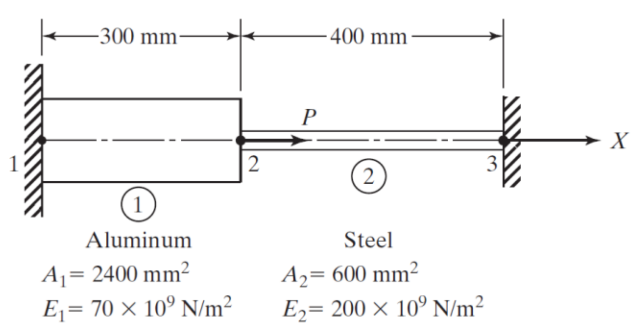


FIGURE E3.4

Solution

(a) The element stiffness matrices are

$$\mathbf{k}^1 = \frac{70 \times 10^3 \times 2400}{300}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad \begin{matrix} 1 & 2 \end{matrix} \leftarrow \text{Global dof}$$

and

$$\mathbf{k}^2 = \frac{200 \times 10^3 \times 600}{400}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \quad \begin{matrix} 2 & 3 \end{matrix}$$

The structural stiffness matrix that is assembled from $\mathbf{k}^1$ and $\mathbf{k}^2$ is

$$\begin{matrix} \quad & 1 & 2 & 3 \end{matrix}$$

$$\mathbf{K} = 10^6\begin{bmatrix} 0.56 & -0.56 & 0 \\ -0.56 & 0.86 & -0.30 \\ 0 & -0.30 & 0.30 \end{bmatrix}$$

The global load vector is

$$\mathbf{F} = [0,\ 200 \times 10^3,\ 0]^{\mathrm{T}}$$

Now dofs 1 and 3 are fixed. When using the penalty approach, therefore, a large number C is added to the first and third diagonal elements of $\mathbf{K}$. Choosing C based on Eq. 3.83, we get

$$C = [0.86 \times 10^6] \times 10^4$$

Thus, the modified stiffness matrix is

$$\mathbf{K} = 10^6\begin{bmatrix} 8600.56 & -0.56 & 0 \\ -0.56 & 0.86 & -0.30 \\ 0 & -0.30 & 8600.30 \end{bmatrix}$$

The finite element equations are given by

$$10^6\begin{bmatrix} 8600.56 & -0.56 & 0 \\ -0.56 & 0.86 & -0.30 \\ 0 & -0.30 & 8600.30 \end{bmatrix}\begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 200 \times 10^3 \\ 0 \end{Bmatrix}$$

which yields the solution

$$\mathbf{Q} = [15.1432 \times 10^{-6}, 0.23257, 8.1127 \times 10^{-6}]^{\mathrm{T}}\ \text{mm}$$

(b) The element stresses (Eq. 3.19) are

$$\sigma_1 = 70 \times 10^3 \times \frac{1}{300}[-1 \quad 1]\begin{Bmatrix} 15.1432 \times 10^{-6} \\ 0.23257 \end{Bmatrix}$$

$$= 54.27 \text{ MPa}$$

where 1 MPa $= 10^6$ N/m^2 $=$ 1 N/mm^2. Also,

$$\sigma_2 = 200 \times 10^3 \times \frac{1}{400}[-1 \quad 1]\begin{Bmatrix} 0.23257 \\ 8.1127 \times 10^{-6} \end{Bmatrix}$$

$$= -116.29 \text{ MPa}$$

(c) The reaction forces are obtained from Eq. 3.78 as

$$\begin{aligned} R_1 &= -CQ_1 \\ &= -[0.86 \times 10^{10}] \times 15.1432 \times 10^{-6} \\ &= -130.23 \times 10^3 \end{aligned}$$

Also,

$$\begin{aligned} R_3 &= -CQ_3 \\ &= -[0.86 \times 10^{10}] \times 8.1127 \times 10^{-6} \\ &= -69.77 \times 10^3 \text{ N} \end{aligned}$$

Example:

In Fig. E3.5a, a load $P = 60 \times 10^3$ N is applied as shown. Determine the displacement field, stress, and support reactions in the body. Take $E = 20 \times 10^3$ N/mm^2.

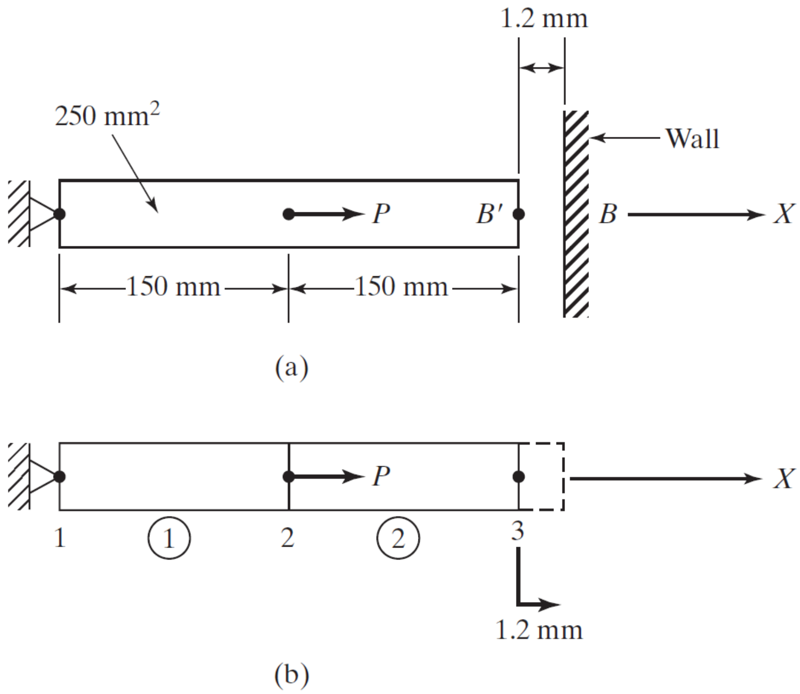


FIGURE E3.5

Solution In this problem, we should first determine whether contact occurs between the bar and the wall, B. To do this, assume that the wall does not exist. Then, the solution to the problem can be verified to be

$$Q_{B'} = 1.8 \text{ mm}$$

where $Q_{B'}$ is the displacement of point B'. From this result, we see that contact does occur. The problem has to be resolved, since the boundary conditions are now different: The displacement at B' is specified to be 1.2 mm. Consider the two-element finite element model in Fig. E3.5b. The boundary conditions are $Q_1 = 0$ and $Q_3 = 1.2$ mm. The structural stiffness matrix **K** is

$$\mathbf{K} = \frac{20 \times 10^3 \times 250}{150}\begin{bmatrix} 1 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

and the global load vector **F** is

$$\mathbf{F} = [0, 60 \times 10^3, 0]^{\mathrm{T}}$$

In the penalty approach, the boundary conditions $Q_1 = 0$ and $Q_3 = 1.2$ imply the following modifications: A large number C chosen here as $C = (2/3) \times 10^9$, is added on to the 1st and 3rd diagonal elements of **K**. Also, the number $(C \times 1.2)$ gets added on to the 3rd component of **F**. Thus, the modified equations are

$$\frac{10^5}{3}\begin{bmatrix} 20001 & -1 & 0 \\ -1 & 2 & -1 \\ 0 & -1 & 20001 \end{bmatrix}\begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 60.0 \times 10^3 \\ 80.0 \times 10^7 \end{Bmatrix}$$

The solution is

$$\mathbf{Q} = [7.49985 \times 10^{-5},\ 1.500045,\ 1.200015]^{\mathrm{T}}\ \mathrm{mm}$$

The element stresses are

$$\sigma_1 = 200 \times 10^3 \times \frac{1}{150}[-1\ \ 1]\begin{Bmatrix} 7.49985 \times 10^{-5} \\ 1.500045 \end{Bmatrix}$$
$$= 199.996\ \mathrm{MPa}$$

$$\sigma_2 = 200 \times 10^3 \times \frac{1}{150}[-1\ \ 1]\begin{Bmatrix} 1.500045 \\ 1.200015 \end{Bmatrix}$$
$$= -40.004\ \mathrm{MPa}$$

The reaction forces are

$$R_1 = -C \times 7.49985 \times 10^{-5}$$
$$= -49.999 \times 10^3\ \mathrm{N}$$

and

$$R_3 = -C \times (1.200015 - 1.2)$$
$$= -10.001 \times 10^3\ \mathrm{N}$$

The results obtained from the penalty approach have a small approximation error due to the flexibility of the support introduced. In fact, the reader may verify that the elimination approach for handling boundary conditions yields the exact reactions, $R_1 = -50.0 \times 10^3$ N and $R_3 = -10.0 \times 10^3$ N. ■

REFERENCES:

1. G. S. Pandit and S.P. Gupta, "Structural Analysis – A Matrix Approach", Second Edition, Mc.Graw Hill Education.
2. S SBhavikatti, "Finite Element Analysis", First Edition, New Age International Pvt. Ltd., Publishers.
3. G. Lakshmi Narasaiah, "Finite Element Analysis", First Edition, BS Publications.
4. Tirupathi R. Chandrupatla and Ashok D. Belegundu, "Introduction to Finite Elements in Engineering", Fourth Edition, Pearson Education Limited.