UNIT - 5

FINITE ELEMENT ANALYSIS

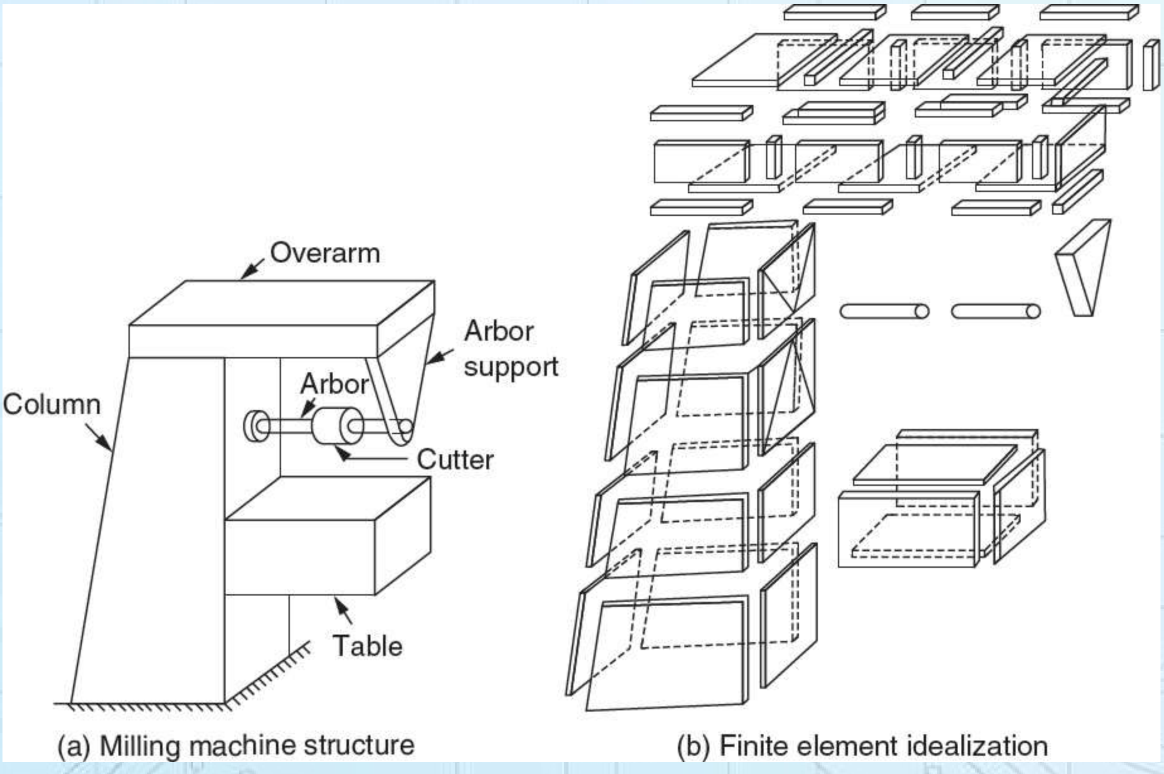
Important Repeated Questions:

1. **Define Finite Element Analysis (FEA) and explain its application/significance in engineering.** (W25 - Q4a, 03 marks) (S25 - Q4a, 03 marks)
2. **Explain the Penalty approach for handling boundary conditions in FEA with an example.** (W25 - Q4c, 07 marks) (S25 - Q4b, 04 marks) (S22 - Q5b OR, 04 marks) (W23 - Q5a, 03 marks)
3. **Determine nodal displacements, stresses, and reaction forces for a given 1-D bar/stepped bar/stepped shaft using FEA (using Penalty or Elimination approach).** (W25 - Q4c, 07 marks) (S23 - Q5c, 07 marks) (S23 - Q5c OR, 07 marks) (S24 - Q4c, 07 marks) (S24 - Q4c OR, 07 marks) (W22 - Q5c, 07 marks) (W23 - Q4c, 07 marks) (S22 - Q5c, 07 marks) (S22 - Q5c OR, 07 marks) (S25 - Q4c, 07 marks) (W24 - Q4c OR, 07 marks)
4. **What is discretization? Explain its concept and importance in FEA.** (W25 - Q3b OR, 04 marks) (S24 - Q4a OR, 03 marks) (S25 - Q4a OR, N/A marks)
5. **Explain the General/Basic steps involved in solving a problem using FEA.** (W25 - Q3a OR, 03 marks) (W23 - Q4b, N/A marks)
6. **Explain/Discuss the properties of the global stiffness matrix.** (W22 - Q5a, 03 marks) (W23 - Q5b, 04 marks) (S22 - Q5a OR, 03 marks) (W24 - Q4a OR, 03 marks)
7. **Define/Optimization in the context of engineering and explain its importance/applications.** (W25 - Q5a, 03 marks) (S24 - Q5b, 04 marks) (W23 - Q4a OR, 03 marks) (S25 - Q5a, 03 marks) (W24 - Q4c OR, 07 marks)
8. **Explain Johnson's method of optimum design.** (W25 - Q5c OR, 07 marks) (W23 - Q5c, 07 marks) (S25 - Q5c, N/A marks)
9. **Explain/Differentiate between constrained and unconstrained optimization.** (W25 - Q5c, 07 marks)
10. **Explain the following with reference to optimization: i) Objective function ii) Constraints.** (W22 - Q4c, 07 marks) (W24 - Q4b, 04 marks)
11. **Explain how the stiffness matrix is derived from the strain-displacement matrix.** (W25 - Q5b OR, 04 marks) (S25 - Q5b, 04 marks)
12. **Explain/Differentiate between Plane stress and Plane strain conditions.** (S23 - Q5b, 04 marks) (W23 - Q4b OR, 04 marks) (W24 - Q4b OR, 04 marks)
13. **Explain the elimination approach used in FEA.** (W22 - Q5b, 04 marks)

Legends: W- Winter, S- Summer, Q- Question and 03/04/07- Marks of Question

5.1 Basic Concept

- The basic idea in the finite element method is to find the solution of a complicated problem by replacing it by a simpler one.
- Since the actual problem is replaced by a simpler one in finding the solution, we will be able to find only an approximate solution rather than the exact solution.
- The existing mathematical tools will not be sufficient to find the exact solution (and sometimes, even an approximate solution) of most of the practical problems.
- Thus, in the absence of any other convenient method to find even the approximate solution of a given problem, we have to prefer the finite element method.
- Moreover, in the finite element method, it will often be possible to improve or refine the approximate solution by spending more computational effort.
- In the finite element method, the solution region is considered as built up of many small, interconnected sub regions called finite elements. As an example of how a finite element model might be used to represent a complex geometrical shape, consider the milling machine structure shown in Figure 5.1(a).
- Since it is very difficult to find the exact response (like stresses and displacements) of the machine under any specified cutting (loading) condition, this structure is approximated as composed of several pieces as shown in Figure 5.1(b) in the finite element method. In each piece or element, a convenient approximate solution is assumed and the conditions of overall equilibrium of the structure are derived. The satisfaction of these conditions will yield an approximate solution for the displacements and stresses.



(a) Milling machine structure (b) Finite element idealization

Fig. 5.1 Representation of a Milling Machine Structure by Finite Elements.

- Figure 5.2 shows the finite element idealization of a fighter aircraft.

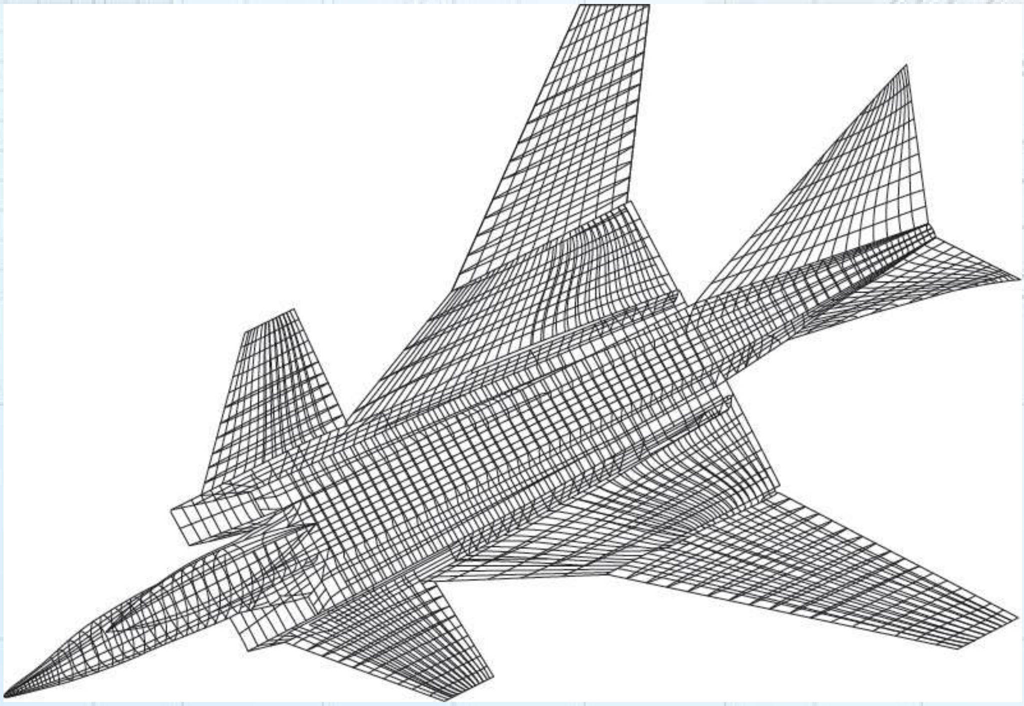
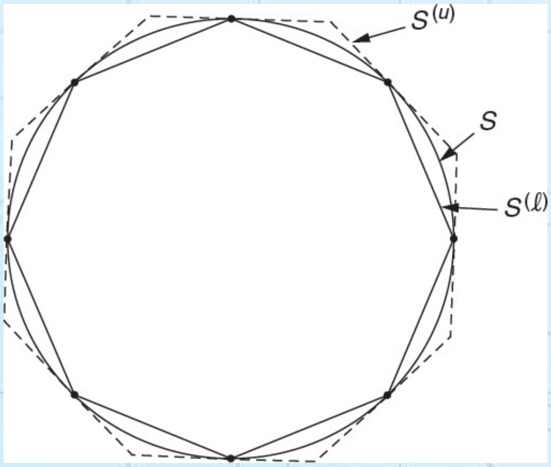


Fig. 5.2 Finite Element Mesh of a Fighter Aircraft.

5.2 Historical Background

- Although the name of the finite element method was given recently, the concept dates back for several centuries. For example, ancient mathematicians found the circumference of a circle by approximating it by the perimeter of a polygon as shown in Figure 5.3.
- In terms of the present day notation, each side of the polygon can be called a “finite element.” By considering the approximating polygon inscribed or circumscribed, one can obtain a lower bound $S^{(l)}$ or an upper bound $S^{(u)}$ for the true circumference S.



- Fig. 5.3 Lower and Upper Bounds to the Circumference of a Circle.

- Furthermore, as the number of sides of the polygon is increased, the approximate values converge to the true value. These characteristics, as will be seen later, will hold true in any general finite element application.
- To find the differential equation of a surface of minimum area bounded by a specified closed curve, Schell back discretized the surface into several triangles and used a finite difference expression to find the total discretized area in 1851.
- In the current finite element method, a differential equation is solved by replacing it by a set of algebraic equations. Since the early 1900s, the behavior of structural frameworks, composed of several bars arranged in a regular pattern, has been approximated by that of an isotropic elastic body.
- In 1943, Courant presented a method of determining the torsional rigidity of a hollow shaft by dividing the cross section into several triangles and using a linear variation of the stress function ϕ over each triangle in terms of the values of ϕ at net points (called nodes in the present day finite element terminology).
- This work is considered by some to be the origin of the present-day finite element method. Since mid-1950s, engineers in aircraft industry have worked on developing approximate methods for the prediction of stresses induced in aircraft wings.
- In 1956, Turner, Cough, Martin, and Topp presented a method for modeling the wing skin using three-node triangles. At about the same time, Argyris and Kelsey presented several papers outlining matrix procedures, which contained some of the finite element ideas, for the solution of structural analysis problems. Reference is considered as one of the key contributions in the development of the finite element method.
- The name finite element was coined, for the first time, by Clough in 1960. Although the finite element method was originally developed mostly based on intuition and physical argument, the method was recognized as a form of the classical Rayleigh-Ritz method in the early 1960s.
- Once the mathematical basis of the method was recognized, the developments of new finite elements for different types of problems and the popularity of the method started to grow almost exponentially.
- The digital computer provided a rapid means of performing the many calculations involved in the finite element analysis and made the method practically viable. Along with the development of high-speed digital computers, the application of the finite element method also progressed at a very impressive rate.
- Zienkiewicz and Cheung presented the broad interpretation of the method and its applicability to any general field problem. The book by Przemieniecki presents the finite element method as applied to the solution of stress analysis problems.

5.3 Definition:

- In Finite Element Analysis, the structure or body is divided into finite numbers of elements, the solution is obtained for individual element and solution of all elements is assembled to give distribution of field variable over entire region.

- For example: Heat Analysis → Field variable is Temperature
  Stress & strain → Field variable is Displacement.

5.4 Engineering Applications of the Finite Element Method

The finite element method was developed originally for the analysis of aircraft structures.

I. Equilibrium problems or steady-state or time-independent problems
   - In an equilibrium problem, we need to find the steady-state displacement or stress distribution if it is a solid mechanics problem,
   - Temperature or heat flux distribution if it is a heat transfer problem and
   - Pressure or velocity distribution if it is a fluid mechanics problem.

II. Eigenvalue problems
   - In eigenvalue problems also, time will not appear explicitly. They may be considered as extensions of equilibrium problems in which critical values of certain parameters are to be determined in addition to the corresponding steady-state configurations.
   - In these problems, we need to find the natural frequencies or buckling loads and mode shapes if it is a solid mechanics or structures problem.
   - Stability of laminar flows if it is a fluid mechanics problem and
   - Resonance characteristics if it is an electrical circuit problem.

III. Propagation or transient problems
   - The propagation or transient problems are time-dependent problems. This type of problem arises, for example, whenever we are interested in finding the response of a body under time-varying force in the area of a solid mechanics
   - Under sudden heating or cooling in the field of heat transfer.
   - Crack propagation.

Engineering Applications of the Finite Element Method

Area of Study	Equilibrium Problems	Eigenvalue Problems	Propagation Problems
1.Civil engineering structures	Static analysis of trusses, frames, folded plates, shell roofs, shear walls, bridges, and prestressed concrete structures	Natural frequencies and modes of structures; stability of structures	Propagation of stress waves; response of structures to a periodic loads
2. Aircraft structures	Static analysis of aircraft wings, fuselages, fins, rockets, spacecraft, and missile structures	Natural frequencies, flutter, and stability of aircraft, rocket, spacecraft, and missile structures	Response of aircraft structures to random loads; dynamic response of aircraft and spacecraft to a periodic loads
3. Heat	Steady-state	–	Transient heat flow in

conduction	temperature distribution in solids and fluids		rocket nozzles, internal combustion engines, turbine blades, fins, and building structures
4. Geo mechanics	Analysis of excavations, retaining walls, underground openings, rock joints, and soil–structure interaction problem; stress analysis in soils, dams, layered piles, and machine foundations	Natural frequencies and modes of dam-reservoir systems and soil–structure interaction problems	Time-dependent soil–structure interaction problems; transient see page in soils and rocks; stress wave propagation in soils and rocks
5. Hydraulic and water resources engineering; hydrodynamics	Analysis of potential flows, free surface flows, boundary layer flows, viscous flows, transonic aerodynamic problems; analysis of hydraulic structures and dams	Natural periods and modes of shallow basins, lakes, and harbors; sloshing of liquids in rigid and flexible containers	Analysis of unsteady fluid flow and wave propagation problems; transient seepage in aquifers and porous media; rarefied gas dynamics; magneto hydrodynamic flows
6. Nuclear engineering	Analysis of nuclear pressure vessels and containment structures; steady –state Temperature distribution in reactor components	Natural frequencies and stability of containment structures; neutron flux distribution	Response of reactor containment structures to dynamic loads; unsteady temperature distribution in reactor components; thermal and viscoelastic analysis of reactor structures
7. Biomedical engineering	Stress analysis of eyeballs, bones, and teeth; load-bearing capacity of implant and prosthetic systems; mechanics of heart valves	–	Impact analysis of skull; dynamics of anatomical structures
8. Mechanical Design	Stress concentration problems; stress analysis of pressure vessels, pistons, composite materials, linkages, and gears	Natural frequencies and stability of linkages, gears, and machine tools	Crack and fracture problems under dynamic loads
9. Electrical machines and electromagnetics	Steady-state analysis of synchronous and induction machines, eddy current, and core losses in electric machines, magneto statics	–	Transient behavior of Electromechanical devices such as motors and actuators, magneto dynamics

5.5 General procedure of Finite Element Method

Step 1: Divide structure into discrete elements (discretization).

- Divide the structure or solution region into subdivisions or elements. Hence, the structure is to be modeled with suitable finite elements.
- The number, type, size, and arrangement of the elements are to be decided.

Step 2: Select a proper interpolation or displacement model.

- Since the displacement solution of a complex structure under any specified load conditions cannot be predicted exactly, we assume some suitable solution within an element to approximate the unknown solution. The assumed solution must be simple from a computational standpoint, but it should satisfy certain convergence requirements.
- In general, the solution or the interpolation model is taken in the form of a polynomial.

Step 3: Derive element stiffness matrices and load vectors.

- From the assumed displacement model finding,

Stiffness matrix – [K]e

Load vector. – [P]e

Step 4: Assemble element equations to obtain the overall equilibrium equations.

- Since the structure is composed of several finite elements, the individual element stiffness matrices and load vectors are to be assembled in a suitable manner and the overall equilibrium equations have to be formulated as

$$[K]\,[\phi] = [P]$$

Where [K] = the assembled stiffness matrix

[ϕ] = the vector of nodal displacements

[P] = Load vector

Step 5: Solve for the unknown nodal displacements.

The overall equilibrium equations have to be modified to account for the boundary conditions of the problem. After the incorporation of the boundary conditions, the equilibrium equations can be expressed as

$$[K]\,[\phi] = [P]$$

Step 6: Compute element strains and stresses.

From the known nodal displacements ϕ, if required, the element strains and stresses can be computed by using the necessary equations of solid or structural mechanics.

5.6 Stress analysis of stepped bar

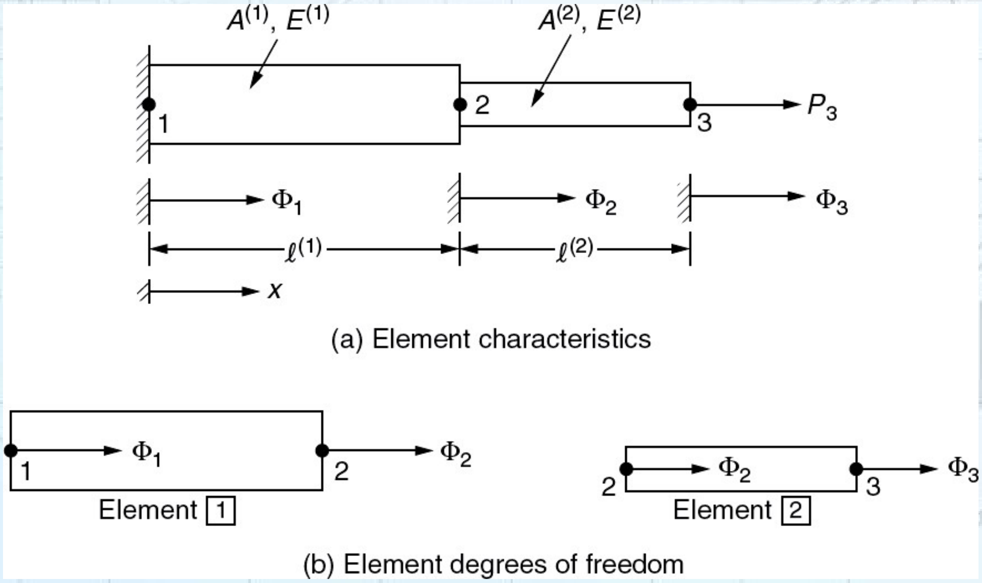


Fig. 5.4 Steeped bar under Axial load

Step 1: Idealization.

The bar is idealized as an assemblage of two elements, one element for each step of the bar as shown in Figure 5.4(b). Each element is assumed to have nodes at the ends so that the stepped bar will have a total of three nodes.

Since the load is applied in the axial direction, the axial displacements of the three nodes are considered as the nodal unknown degrees of freedom of the system, and are denoted as Φ_1,Φ_2, and Φ_3 as shown in Figure 5.4(b).

Step 2: Develop interpolation or displacement model.

Since the two end displacements of element *e*, $\Phi_1^{(e)}$, and $\Phi_2^{(e)}$, are considered the degrees of freedom, the axial displacement, $\phi(x)$, within the element *e* is assumed to vary linearly as (Figure 5.4(c)): $\phi(x) = a+bx$ (E.1)

where a and b are constants that can be expressed in terms of the end (nodal) displacements of the element $\Phi_1^{(e)}$ and $\Phi_2^{(e)}$, as follows. Since $\phi(x)$ must be equal to $\Phi_1^{(e)}$ at $x = 0$ and $\Phi_2^{(e)}$, at $x = l^{(e)}$, we obtain

$$\phi(x=0) = a \equiv \Phi_1^{(e)},\ \phi(x=l^{(e)}) = a + bl^{(e)} = \Phi_2^{(e)} \qquad \text{(E.2)}$$

Equations (E.2) yield the solution

$$a = \Phi_1^{(e)},\ b = \left(\frac{\Phi_2^{(e)} - \Phi_1^{(e)}}{l^{(e)}}\right) \qquad \text{(E.3)}$$

Thus the axial displacement of the element e, Eq. (E.1), can be expressed as

$$\phi(x) = \Phi_1^{(e)} + \left(\frac{\Phi_2^{(e)} - \Phi_1^{(e)}}{l^{(e)}}\right)x \qquad \text{(E.4)}$$

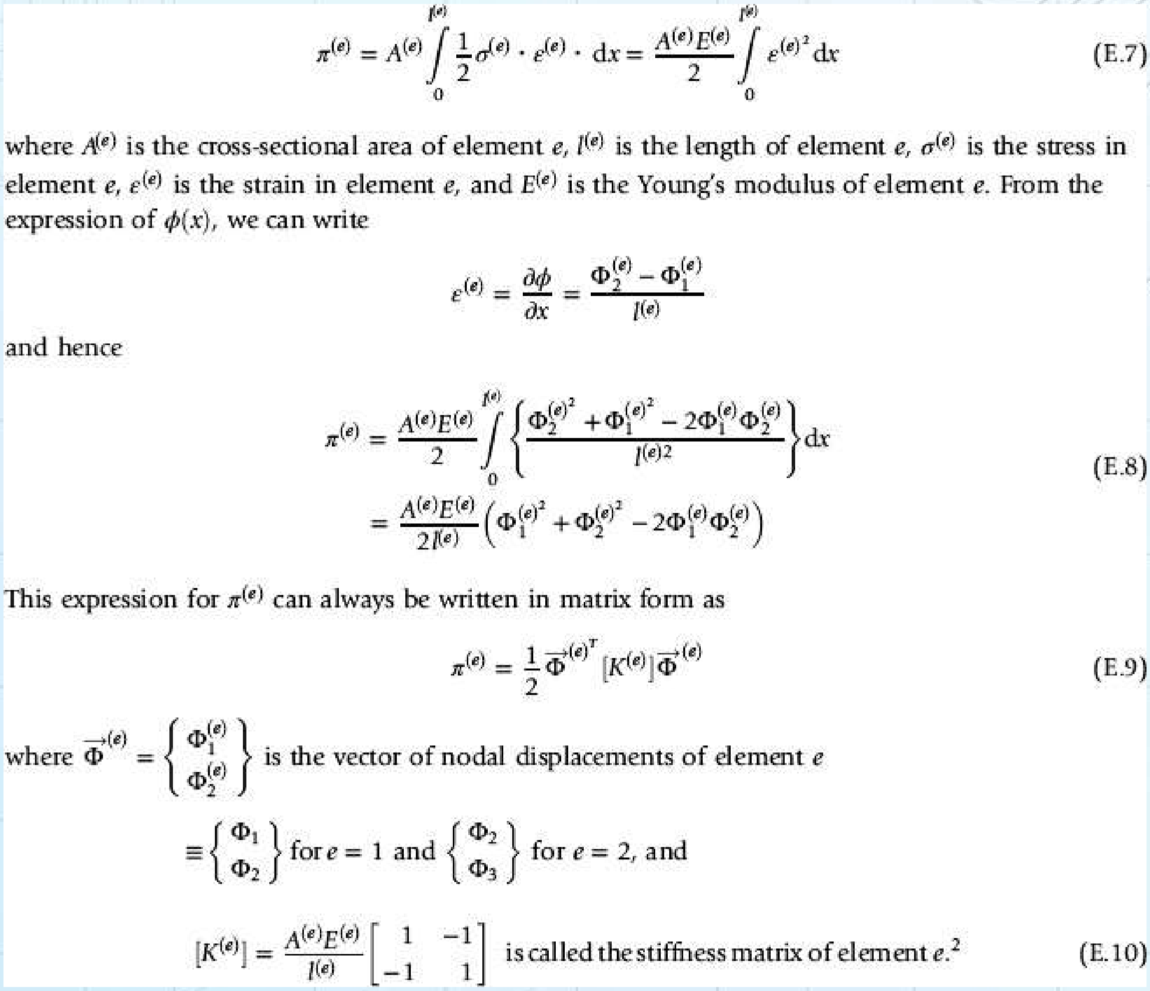
Step 3: Derive element stiffness matrix and element load vector.

The element stiffness matrices can be derived from the principle of minimum potential energy. For this, we write the potential energy of the bar (I) under axial deformation as

I = strain energy – work done by external forces

$= \square^{(1)} + \square^{(2)} - Wp$

where $\square^{(e)}$ represents the strain energy of element *e*, and Wp denotes the work done by external forces acting on the bar.



$$\pi^{(e)} = A^{(e)} \int_0^{l^{(e)}} \frac{1}{2}\sigma^{(e)} \cdot \varepsilon^{(e)} \cdot \mathrm{d}x = \frac{A^{(e)}E^{(e)}}{2} \int_0^{l^{(e)}} \varepsilon^{(e)^2} \mathrm{d}x \tag{E.7}$$

where $A^{(e)}$ is the cross-sectional area of element e, $l^{(e)}$ is the length of element e, $\sigma^{(e)}$ is the stress in element e, $\varepsilon^{(e)}$ is the strain in element e, and $E^{(e)}$ is the Young's modulus of element e. From the expression of $\phi(x)$, we can write

$$\varepsilon^{(e)} = \frac{\partial \phi}{\partial x} = \frac{\Phi_2^{(e)} - \Phi_1^{(e)}}{l^{(e)}}$$

and hence

$$\begin{aligned} \pi^{(e)} &= \frac{A^{(e)}E^{(e)}}{2} \int_0^{l^{(e)}} \left\{ \frac{\Phi_2^{(e)^2} + \Phi_1^{(e)^2} - 2\Phi_1^{(e)}\Phi_2^{(e)}}{l^{(e)2}} \right\} \mathrm{d}x \\ &= \frac{A^{(e)}E^{(e)}}{2l^{(e)}} \left(\Phi_1^{(e)^2} + \Phi_2^{(e)^2} - 2\Phi_1^{(e)}\Phi_2^{(e)} \right) \end{aligned} \tag{E.8}$$

This expression for $\pi^{(e)}$ can always be written in matrix form as

$$\pi^{(e)} = \frac{1}{2} \vec{\Phi}^{(e)^T} [K^{(e)}] \vec{\Phi}^{(e)} \tag{E.9}$$

where $\vec{\Phi}^{(e)} = \begin{Bmatrix} \Phi_1^{(e)} \\ \Phi_2^{(e)} \end{Bmatrix}$ is the vector of nodal displacements of element e

$\equiv \begin{Bmatrix} \Phi_1 \\ \Phi_2 \end{Bmatrix}$ for $e = 1$ and $\begin{Bmatrix} \Phi_2 \\ \Phi_3 \end{Bmatrix}$ for $e = 2$, and

$$[K^{(e)}] = \frac{A^{(e)}E^{(e)}}{l^{(e)}} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{E.10}$$

is called the stiffness matrix of element e.[2]

Apply principle of potential energy

Since there are only concentrated loads acting at the nodes of the bar (and no distributed load acts on the bar), the work done by external forces can be expressed as

$Wp = \Phi_1 P_1 + \Phi_2 P_2 + \Phi_3 P_3$

In the present case, $\Phi_1 = 0$ since node 1 is fixed while the loads applied externally at the nodes 1, 2, and 3 in the directions of Φ_1, Φ_2, and Φ_3, respectively, are P_1 = unknown (denotes the reaction at the fixed node 1 where the displacement Φ_1 is zero), $P_2 = 0$, and $P_3 = 1$ N.

If the bar as a whole is in equilibrium under the loads, $\vec{P} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix}$ the principle of minimum potential energy gives

$$\frac{\partial I}{\partial \Phi_i} = 0, \quad i = 1, 2, 3$$

This equation can be rewritten as

$$\frac{\partial I}{\partial \Phi_i} = \frac{\partial}{\partial \Phi_i}\left(\sum_{e=1}^{2} \pi^{(e)} - W_p\right) = 0, \quad i = 1, 2, 3$$

Step 4: Assemble element stiffness matrices and element load vectors

This step includes the assembly of element stiffness matrices [K]e and element load vectors [P]e toobtain the overall or global equilibrium equations.

[K] [ϕ] = [P]

where[K] is the assembled or global stiffness matrix = $\sum_{e=1} [K^{(e)}]$

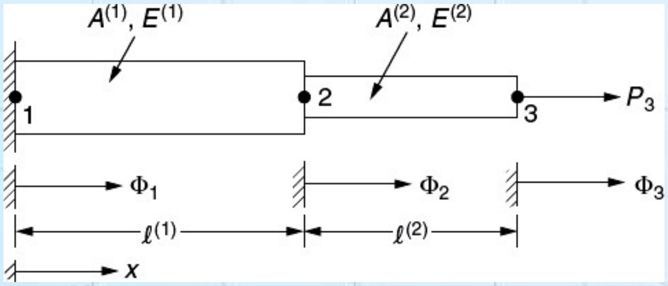
$$\vec{\Phi} = \begin{Bmatrix} \Phi_1 \\ \Phi_2 \\ \Phi_3 \end{Bmatrix}$$ = vector ofglobal displacements.

Example 5.1: $A_1 = 200$ mm^2, $A_2 = 100$ mm^2, $P_3 = 1000$ N.

$E_1 = E_2 = E = 2 \times 10^6$ N/mm^2

$l_1 = l_2 = 100$ mm

Find: Displacement and stress & strain.



$$[K^{(1)}] = \frac{A^{(1)}E^{(1)}}{l^{(1)}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^6 \begin{array}{c} \begin{array}{cc} \Phi_1 & \Phi_2 \end{array} \\ \begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix} \end{array} \begin{array}{c} \\ \Phi_1 \\ \Phi_2 \end{array} \tag{E.16}$$

$$[K^{(2)}] = \frac{A^{(2)}E^{(2)}}{l^{(2)}}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^6 \begin{array}{c} \begin{array}{cc} \Phi_2 & \Phi_3 \end{array} \\ \begin{bmatrix} 2 & -2 \\ -2 & 2 \end{bmatrix} \end{array} \begin{array}{c} \\ \Phi_2 \\ \Phi_3 \end{array} \tag{E.17}$$

Let overall stiffness matrix [K] = [K$^{(1)}$] + [K$^{(2)}$]

$$[K] = 10^6 \begin{array}{c} \\ \\ \\ \end{array}\begin{array}{ccc|c} \Phi_1 & \Phi_2 & \Phi_3 & \\ 4 & -4 & 0 & \Phi_1 \\ -4 & 4+2 & -2 & \Phi_2 \\ 0 & -2 & 2 & \Phi_3 \end{array} = 2\times 10^6 \begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix}$$

In the present case, external loads act only at the node points; as such, there is no need to assemblethe element load vectors. The overall or global load vector can be written as

$$\vec{P} = \begin{Bmatrix} P_1 \\ P_2 \\ P_3 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ 0 \\ 1 \end{Bmatrix}$$

$$2\times 10^6 = \begin{bmatrix} 2 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} \Phi_1 \\ \Phi_2 \\ \Phi_3 \end{Bmatrix} = \begin{Bmatrix} P_1 \\ 0 \\ 1 \end{Bmatrix}$$

$$2\times 10^6 \begin{bmatrix} 3 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} \Phi_2 \\ \Phi_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 1 \end{Bmatrix}$$

By solving the matrix

$\Phi_2 = 0.25 \times 10^{-6}$cm and

$\Phi_3 = 0.75 \times 10^{-6}$cm

Derive element strains and stresses.

Once the displacements are computed, the strains in the elements can be found as

$$\varepsilon^{(1)} = \frac{\partial \phi}{\partial x} \text{ for element 1} = \frac{\Phi_2^{(1)} - \Phi_1^{(1)}}{l^{(1)}} \equiv \frac{\Phi_2 - \Phi_1}{l^{(1)}} = 0.25 \times 10^{-7}$$

$$\varepsilon^{(2)} = \frac{\partial \phi}{\partial x} \text{ for element 2} = \frac{\Phi_2^{(2)} - \Phi_1^{(2)}}{l^{(2)}} \equiv \frac{\Phi_3 - \Phi_2}{l^{(2)}} = 0.50 \times 10^{-7}$$

The stresses in the elements are given by

$$\sigma^{(1)} = E^{(1)}\varepsilon^{(1)} = \left(2\times 10^7\right)\left(0.25\times 10^{-7}\right) = 0.5 \text{ N/cm}^2$$

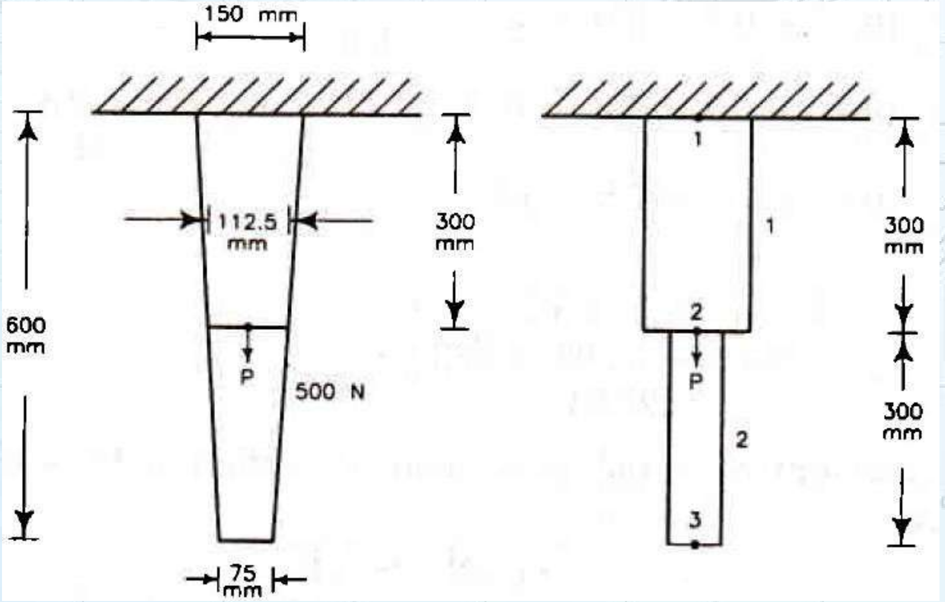
$$\sigma^{(2)} = E^{(2)}\varepsilon^{(2)} = \left(2\times 10^7\right)\left(0.50\times 10^{-7}\right) = 1.0 \text{ N/cm}^2$$

Example 5.2: A thin plate as shown in Fig. 5.5(a) has uniform thickness of 2 cm and its modulus of elasticity is 200 x 10^3 N/mm^2 and density 7800 kg/m^3. In addition to its self weight the plate is subjected to a point load P of500 N is applied at its midpoint.
Solve the following:

(i) Finite element model with two finite elements.
(ii) Global stiffness matrix.
(iii) Global load matrix.
(iv) Displacement at nodal point.

(v) Stresses in each element.

(vi) Reaction at support.



(a) Fig. 5.5 (b)

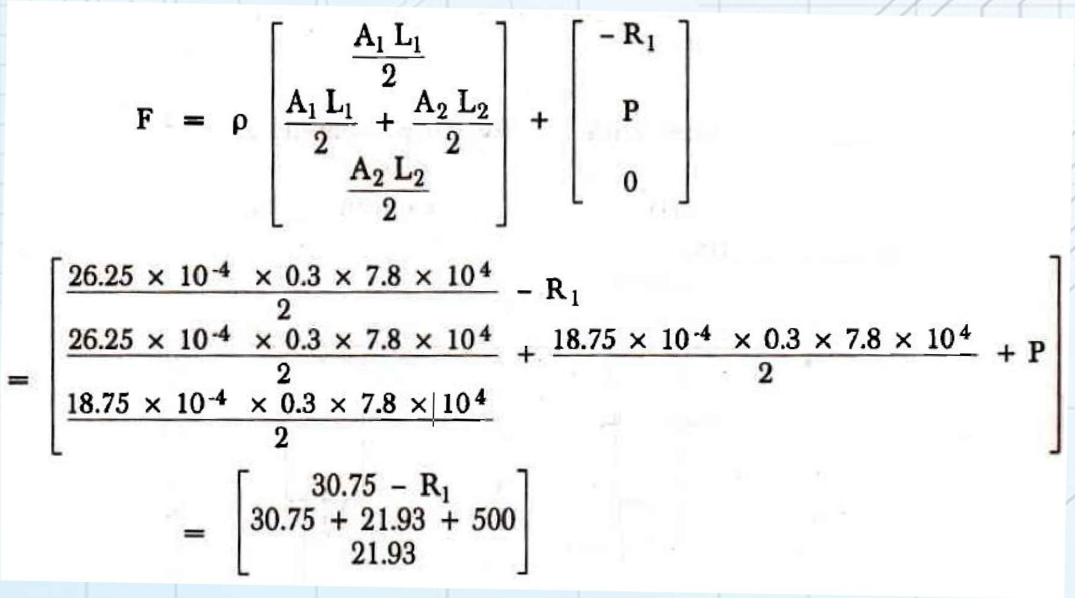
(i) The tapered plate can be idealized as two element model with the tapered area converted to the rectangular equivalent area Refer Fig. (b). The areas A_1 and A_2 are equivalent areas calculated as

$$A_1 = \frac{15 + 11.25}{2} \times 2 = 26.25\ cm^2$$

$$A_2 = \frac{11.25 + 7.5}{2} \times 2 = 18.75\ cm^2$$

(ii) Global stiffness matrix can be obtained as

(iii) The load matrix given by

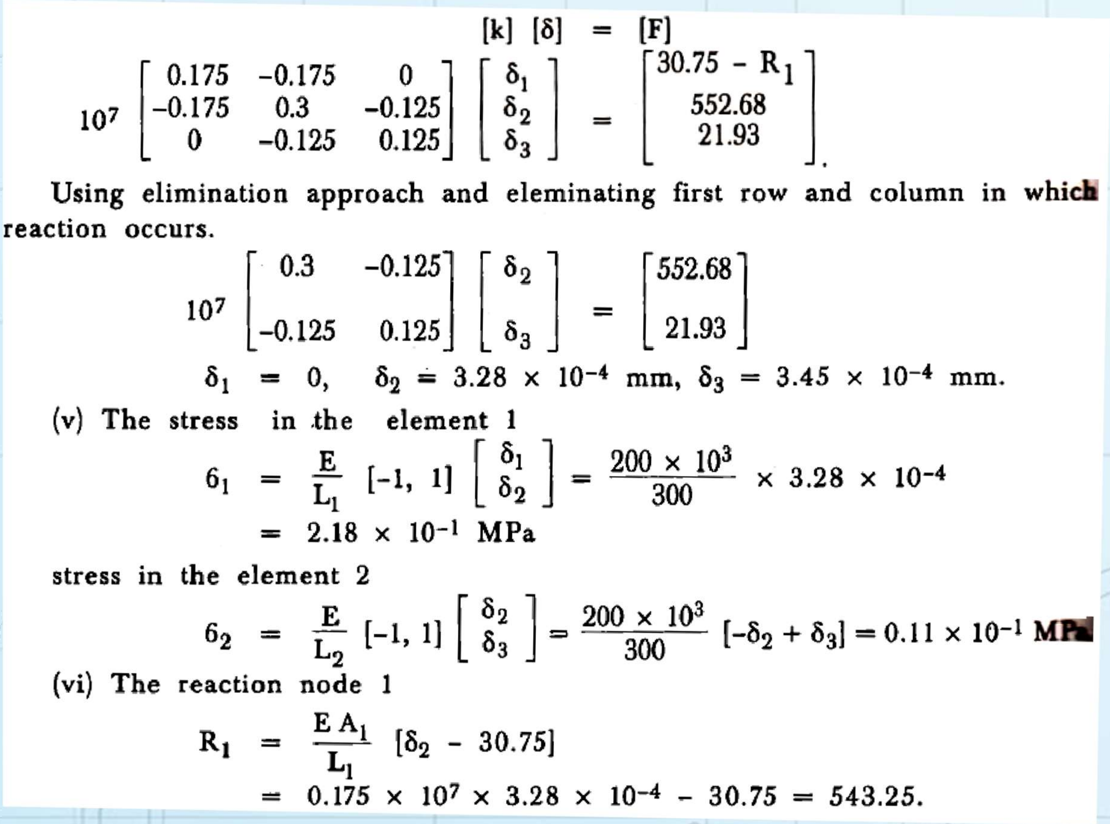


$$F = \rho \begin{bmatrix} \dfrac{A_1 L_1}{2} \\ \dfrac{A_1 L_1}{2} + \dfrac{A_2 L_2}{2} \\ \dfrac{A_2 L_2}{2} \end{bmatrix} + \begin{bmatrix} -R_1 \\ P \\ 0 \end{bmatrix}$$

$$= \begin{bmatrix} \dfrac{26.25 \times 10^{-4} \times 0.3 \times 7.8 \times 10^4}{2} - R_1 \\ \dfrac{26.25 \times 10^{-4} \times 0.3 \times 7.8 \times 10^4}{2} + \dfrac{18.75 \times 10^{-4} \times 0.3 \times 7.8 \times 10^4}{2} + P \\ \dfrac{18.75 \times 10^{-4} \times 0.3 \times 7.8 \times 10^4}{2} \end{bmatrix}$$

$$= \begin{bmatrix} 30.75 - R_1 \\ 30.75 + 21.93 + 500 \\ 21.93 \end{bmatrix}$$

(iv) The displacement at nodal point can be obtained by writing the equation in global form as



$$[k]\ [\delta] = [F]$$

$$10^7 \begin{bmatrix} 0.175 & -0.175 & 0 \\ -0.175 & 0.3 & -0.125 \\ 0 & -0.125 & 0.125 \end{bmatrix} \begin{bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{bmatrix} = \begin{bmatrix} 30.75 - R_1 \\ 552.68 \\ 21.93 \end{bmatrix}.$$

Using elimination approach and eleminating first row and column in which reaction occurs.

$$10^7 \begin{bmatrix} 0.3 & -0.125 \\ -0.125 & 0.125 \end{bmatrix} \begin{bmatrix} \delta_2 \\ \delta_3 \end{bmatrix} = \begin{bmatrix} 552.68 \\ 21.93 \end{bmatrix}$$

$$\delta_1 = 0, \quad \delta_2 = 3.28 \times 10^{-4} \text{ mm}, \quad \delta_3 = 3.45 \times 10^{-4} \text{ mm}.$$

(v) The stress in the element 1

$$\sigma_1 = \frac{E}{L_1} [-1,\ 1] \begin{bmatrix} \delta_1 \\ \delta_2 \end{bmatrix} = \frac{200 \times 10^3}{300} \times 3.28 \times 10^{-4}$$

$$= 2.18 \times 10^{-1} \text{ MPa}$$

stress in the element 2

$$\sigma_2 = \frac{E}{L_2} [-1,\ 1] \begin{bmatrix} \delta_2 \\ \delta_3 \end{bmatrix} = \frac{200 \times 10^3}{300} [-\delta_2 + \delta_3] = 0.11 \times 10^{-1} \text{ MPa}$$

(vi) The reaction node 1

$$R_1 = \frac{E A_1}{L_1} [\delta_2 - 30.75]$$

$$= 0.175 \times 10^7 \times 3.28 \times 10^{-4} - 30.75 = 543.25.$$

5.7 Penalty Approach

- In the preceding problems, the elimination approach was used to achieve simplified matrices. This method though simple, is not very easy to adapt in terms of algorithms written fix computer programs.
- An alternate method to achieve solutions is by the penalty approach. By this approach a rigid support is considered as a spring having infinite stiffness. Consider a system as shown in Fig. 7.6.

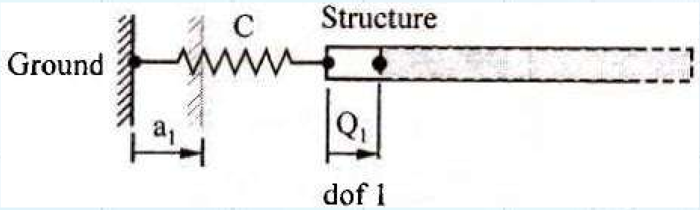


Fig. 5.6 Penalty Approach

- The support or the ground is modelled with a high stiffness spring, having a stiffness C. To represent a rigid ground, c must be infinity.
- However, instead of introducing an infinite value in the calculations, a substantially high value of stiffness constant is introduced for those nodes resting on rigid supports.
- The magnitude of the stiffness constant should be at least 10^4 times more than the maximum value in the global stiffness matrix.
- From Fig. 5.6, it is seen that one end of the spring will displace by a_1. The displacement Q_1 (for dof l) will be approximately equal to a_1 as the spring has a high stiffness.
- Consider a simple 1D element with node 1 fixed.

$$\mathbf{KQ} = \mathbf{F}$$

$$\begin{bmatrix} k_{11} & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \end{Bmatrix}$$

- At node l, the stiffness term is ‘C’ is introduced to reflect the boundary condition related to a rigid support. To compensate this change, the force term will also be modified as:

$$\begin{bmatrix} k_{11}+C & k_{12} \\ k_{21} & k_{22} \end{bmatrix} \begin{Bmatrix} Q_1 \\ Q_2 \end{Bmatrix} = \begin{Bmatrix} F_1+Ca_1 \\ F_2 \end{Bmatrix}$$

- The reaction force as per penalty approach would be found by multiplying the added stiffness with the net deflection of the node.

$$R = - C(Q-a)$$

- The penalty approach is an approximate method and the accuracy of the forces depends on the value of C.

Example 5.3: Consider the bar shown in Fig. 5.7. An axial load P = 200 x 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.

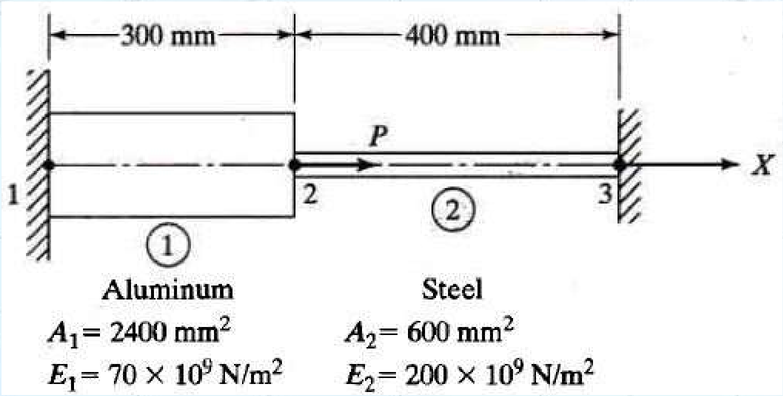


Fig. 5.7

(a) The element stiffness matrices are

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

and

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

The structural stiffness matrix that is assembled from k^1 and k^2 is

$$\mathbf{K} = 10^6 \begin{array}{c} \begin{matrix} \;1 & \;\;\;\;\;2 & \;\;\;\;\;3 \end{matrix} \\ \begin{bmatrix} 0.56 & -0.56 & 0 \\ -0.56 & 0.86 & -0.30 \\ 0 & -0.30 & 0.30 \end{bmatrix} \end{array}$$

The global load vector is

$$F = [0, 200 \text{ x } 10^3, 0]^T$$

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 K. Choosing C

$$C = [0.86 \text{ x } 10^6] \text{ x } 10^4$$

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

$$Q = [15.1432 \text{ x } 10^{-6},\ 0.23257,\ 8.1127 \text{ x } 10^{-6}]\text{mm}$$

(b) The element stresses are

$$\sigma_1 = 70 \times 10^3 \times \frac{1}{300}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} 15.1432 \times 10^{-6} \\ 0.23257 \end{Bmatrix}$$
$$= 54.27 \text{ MPa}$$

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

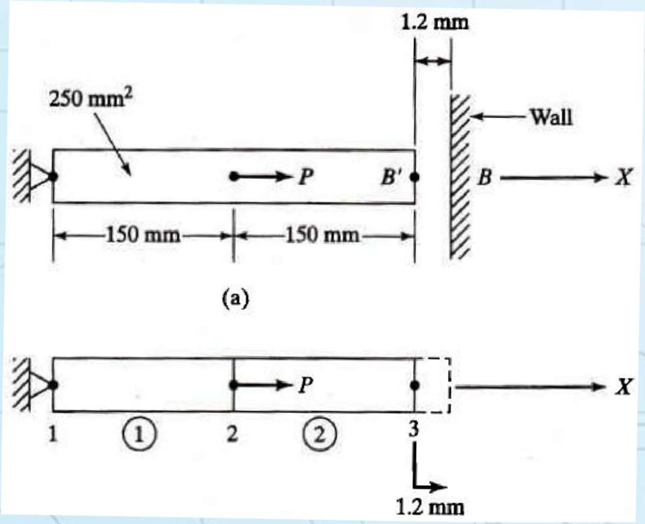
$$\sigma_2 = 200 \times 10^3 \times \frac{1}{400}\begin{bmatrix} -1 & 1 \end{bmatrix}\begin{Bmatrix} 0.23257 \\ 8.1127 \times 10^{-6} \end{Bmatrix}$$
$$= -116.29 \text{ MPa}$$

(c) The reaction forces are

$$R_1 =– CQ_1$$
$$= –[0.86 \text{ x } 10^{10}] \text{ x } 15.1432\text{x } 10^6$$
$$= – 130.23 \text{ x } 10^3 \text{ N}$$

$$R_3 =– CQ_3$$
$$= –[0.86 \text{ x } 10^{10}] \text{ x } 8.1127\text{x } 10^6$$
$$= – 69.77 \text{ x } 10^3 \text{ N}$$

Example 5.4: In Fig. 5.8(a), a load P = 60 x 10^3 N is applied as shown. Determine the displacement field, stress and support reactions in the body. Take E = 20 x 10^3N/mm^2.



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

$$F = [0, 60 \text{ x } 10^3, 0]^T$$

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 $C = (2/3) \text{ x } 10^{10}$, is added on to the 1st and 3rd diagonal elements of K. Also, the number (C x 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

$$Q = [7.49985 \text{ x } 10^{-5}, 1.500045, 1.200015]^T \text{ mm}$$

The element stresses are

$$\begin{aligned} \sigma_1 &= 200 \times 10^3 \times \frac{1}{150}[-1 \quad 1]\begin{Bmatrix} 7.49985 \times 10^{-5} \\ 1.500045 \end{Bmatrix} \\ &= 199.996 \text{ MPa} \\ \sigma_2 &= 200 \times 10^3 \times \frac{1}{150}[-1 \quad 1]\begin{Bmatrix} 1.500045 \\ 1.200015 \end{Bmatrix} \\ &= -40.004 \text{ MPa} \end{aligned}$$

The reaction forces are

$$\begin{aligned} R_1 &= -C \text{x } 7.49985 \text{ x } 10^{-5} \\ &= -49.999 \text{ x } 10^3 \text{N} \\ R_3 &= -C\text{x}(1.200015 - 1.2) \\ &= -10.001 \text{ x } 10^3 \text{N} \end{aligned}$$

5.8Effect of Temperature on Elements:

When any material is subjected to a thermal stress, the thermal load is additional load acting on every element. This load can be calculated by using thermal expansion of the material due to the rise in. temperature.

Thermal stress in material can be given by

$$\sigma_t = E\varepsilon_t$$

Where ε_t= thermal strain

E = modulus of elasticity

$\varepsilon_t = \alpha \, \Delta t$

α= coefficient of linear expansion of material

Δt= change in temperature of material.

Then the thermal load is given by

Where, $F_t = \sigma_t A = AE\alpha\,\Delta t$

A = Area of the bar.

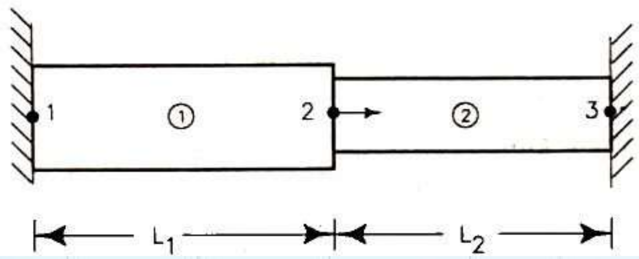


Fig. 5.9

Consider the horizontal step bar supported at two ends is subjected to a thermal stress and load P at node 2 as shown in Fig. 5.9.

Thermal load in element 1

$$[F_1] = \begin{bmatrix} Ft_1 \\ Ft_{12} \\ 0 \end{bmatrix} = A_1\ E\ \alpha\ \Delta t \begin{bmatrix} -1 \\ 1 \\ 0 \end{bmatrix}$$

Thermal load in element 2

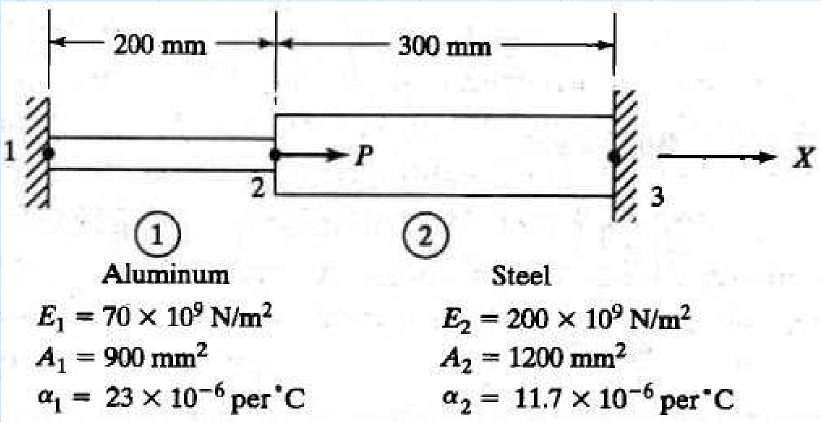
$$[F_2] = \begin{bmatrix} 0 \\ Ft_{21} \\ Ft_3 \end{bmatrix} = A_2\ E\ \alpha\ \Delta t \begin{bmatrix} 0 \\ -1 \\ 1 \end{bmatrix}$$

$$[F] = [F_1] + [F_2] + \begin{bmatrix} 0 \\ P \\ 0 \end{bmatrix} = \begin{bmatrix} -A_1E\alpha\Delta t \\ A_1E\alpha\Delta t - A_2E\alpha\Delta t + P \\ A_2E\alpha\Delta t \end{bmatrix}$$

Example 5.5 : An axial load P = 300 x 10^3 N is applied at 20□C to the rod as shown in Fig. 5.10.The temperature is then raised to 60□C.

(a) Assemble the K and F matrices.

(b) Determine the nodal displacements and element stresses.



(a) The element stiffness matrices are

$$\mathbf{k}^1 = \frac{70 \times 10^3 \times 900}{200}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \text{N/mm}$$

$$\mathbf{k}^2 = \frac{200 \times 10^3 \times 1200}{300}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \text{N/mm}$$

$$\mathbf{K} = 10^3\begin{bmatrix} 315 & -315 & 0 \\ -315 & 1115 & -800 \\ 0 & -800 & 800 \end{bmatrix} \text{N/mm}$$

Now, in assembling F, both temperature and point load effects have to be considered.

The element temperature forces due to $\Delta T = 40°C$ are obtained as

$$\mathbf{\Theta}^1 = 70 \times 10^3 \times 900 \times 23 \times 10^{-6} \times 40\begin{Bmatrix} -1 \\ 1 \end{Bmatrix}\begin{matrix} \overset{\downarrow \text{ Global dof}}{1} \\ 2 \end{matrix} \quad \text{N}$$

$$\mathbf{\Theta}^2 = 200 \times 10^3 \times 1200 \times 11.7 \times 10^{-6} \times 40\begin{Bmatrix} -1 \\ 1 \end{Bmatrix}\begin{matrix} 2 \\ 3 \end{matrix} \quad \text{N}$$

Upon assembling $\mathbf{\Theta}^1$, $\mathbf{\Theta}^2$, and the point load, we get

$$\mathbf{F} = 10^3\begin{Bmatrix} -57.96 \\ 57.96 - 112.32 + 300 \\ 112.32 \end{Bmatrix}$$

$$F = 10^3[-57.96, 245.64, 112.32]^T \text{ N}$$

(b) The elimination approach will now be used to solve for the displacements. Since dofs 1 and 3 are fixed, the first and third rows and columns of K, together with the first and third components of F, are deleted. This results in the scalar equation

$10^3[1115]\, Q_2 = 10^3 \times 245.64$

$Q_2 = 0.220$ mm

$Q = [0, 0.220, 0]^T$mm

In evaluating element stresses

$$\sigma_1 = \frac{70 \times 10^3}{200}[-1 \quad 1]\begin{Bmatrix} 0 \\ 0.220 \end{Bmatrix} - 70 \times 10^3 \times 23 \times 10^{-6} \times 40$$
$$= 12.60 \text{ MPa}$$

$$\sigma_2 = \frac{200 \times 10^3}{300}[-1 \quad 1]\begin{Bmatrix} 0.220 \\ 0 \end{Bmatrix} - 200 \times 10^3 \times 11.7 \times 10^{-6} \times 40$$
$$= -240.27 \text{ MPa}$$

5.9 Discretization of the Domain

- The geometry (domain or solution region) of the problem is often irregular. The first step of the finite element analysis involves the discretization of the irregular domain into smaller and regular subdomains, known as finite elements. This is equivalent to replacing the domain having an infinite number of degrees of freedom (DOF) by a system having a finite number of DOF.
- A variety of methods can be used to model a domain with finite elements.
- Different methods of dividing the domain into finite elements involve varying amounts of computational time and often lead to different approximations to the solution of the physical problem.
- Some automatic mesh generation programs have been developed for the efficient idealization of complex domains requiring minimal interface with the analyst.

- **Basic Element Shapes**

- The shapes, sizes, number, and configurations of the elements have to be chosen carefully such that the original body or domain is simulated as closely as possible without increasing the computational effort needed for the solution.

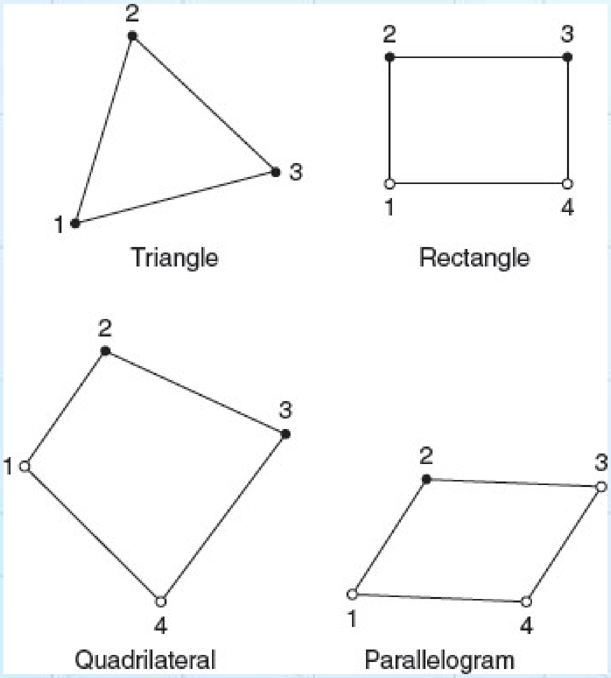


Fig. 5.11 Two - Dimensional Elements.

- The two-dimensional elements shown in Fig. 5.11. The basic element useful for two-dimensional analysis is the triangular element. Although a quadrilateral element (or its special forms, the rectangle and parallelogram) can be obtained by assembling two or four triangular elements, as shown in Fig.5.12.
- For the bending analysis of plates, multiple dof (transverse displacement and its derivatives) are used at each node.

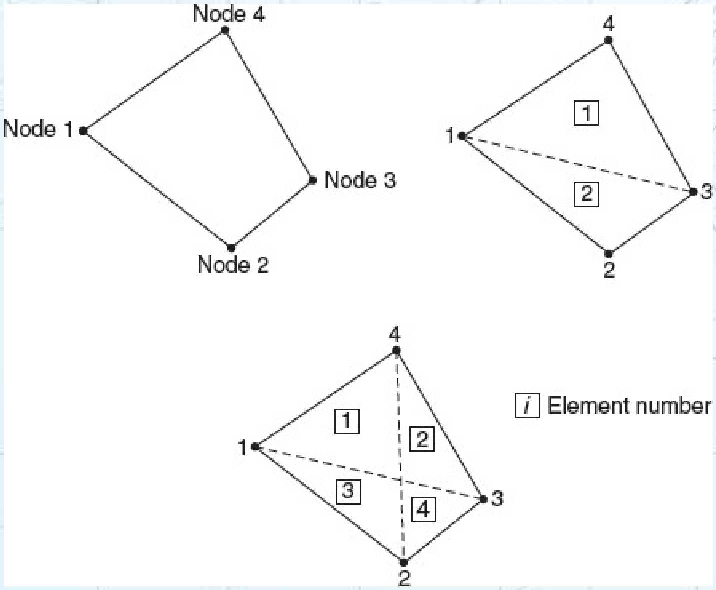


Fig. 5.12 A Quadrilateral Element as an Assemblage of Two or Four Triangular Elements.

- If the geometry, material properties, and other parameters of the body can be described by three independent spatial coordinates, we can idealize the body by using the three-dimensional elements shown in Fig.5.13.

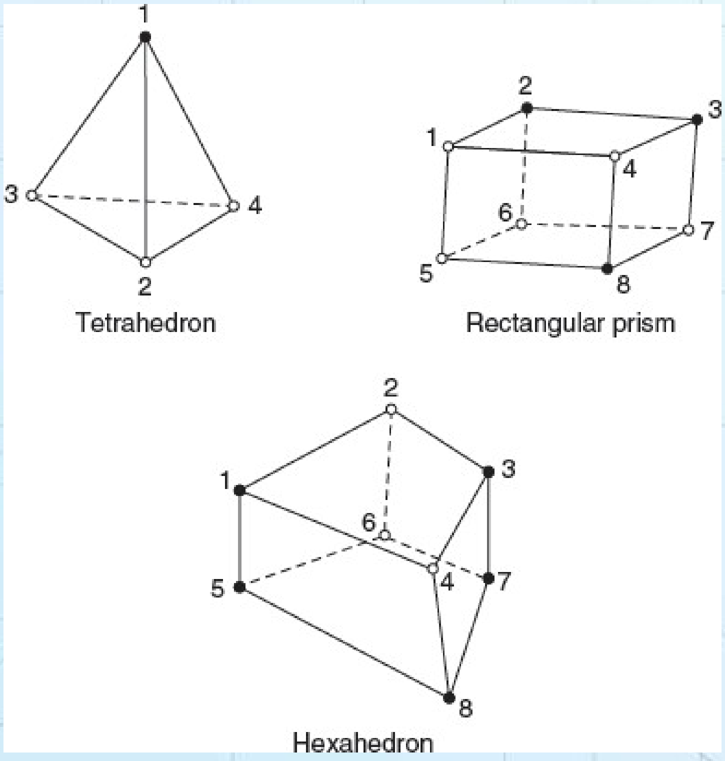


Fig. 5.13Three-Dimensional Finite Elements.

- The basic three-dimensional element, analogous to the triangular element in the case of two-dimensional problems, is the tetrahedron element.

- In some cases the hexahedron element, which can be obtained by assembling five tetrahedrons as indicated in Fig. 5.14.

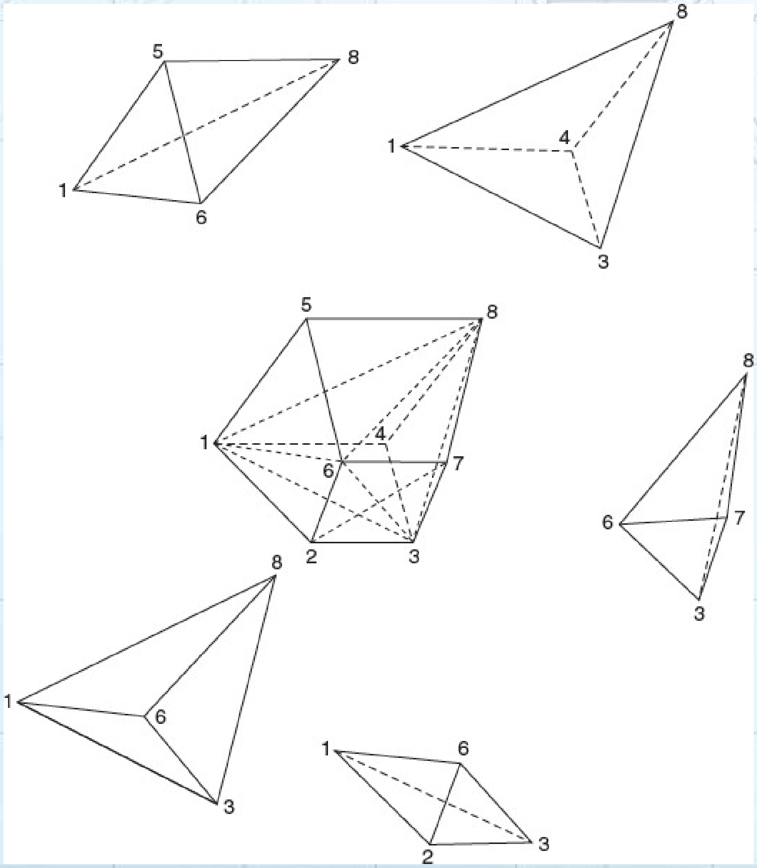
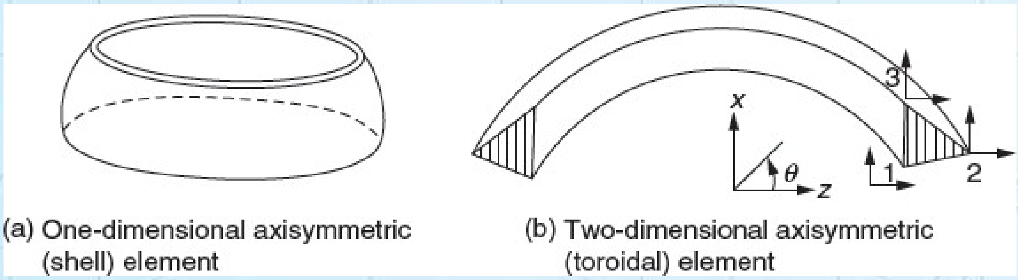


Fig. 5.14 A Hexahedron Element as an Assemblage of Five Tetrahedron Elements.

- Some problems, which are actually three-dimensional, can be described by only one or two independent coordinates. Such problems can be idealized by using an axisymmetric or ring type of elements shown in Fig. 5.15.



(a) One-dimensional axisymmetric (shell) element

(b) Two-dimensional axisymmetric (toroidal) element

Fig. 5.15 Axisymmetric Elements.

- The problems that possess axial symmetry, such as pistons, storage tanks, valves, rocket nozzles, and reentry vehicle heat shields, fall into this category.
- For the discretization of problems involving curved geometries, finite elements with curved sides are useful. Typical elements having curved boundaries are shown in Fig. 5.16.

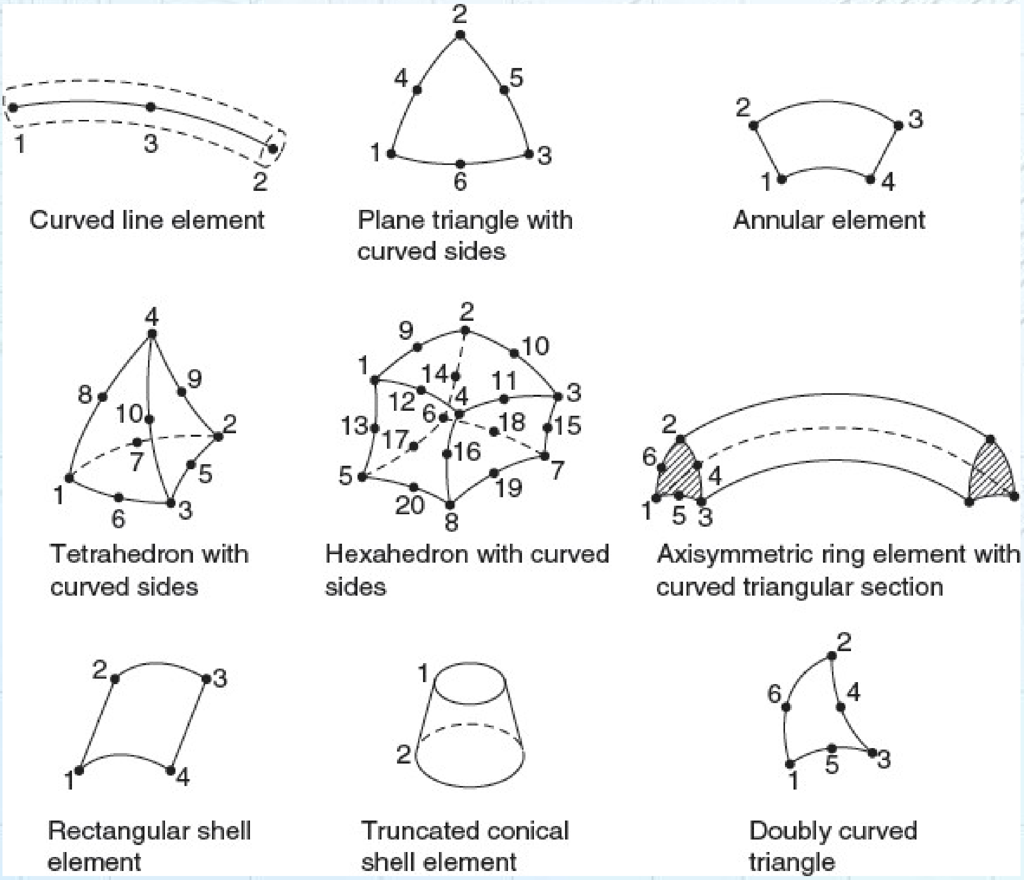


Fig. 5.16 Finite Elements with Curved Boundaries.

5.10Discretization Process

Step - 1: Type of Elements

- If the problem involves the analysis of a truss structure under a given set of load conditions (Fig. 5.17(a)), the type of elements to be used for idealization is obviously the "bar or line elements" as shown in Fig. 5.17(b).

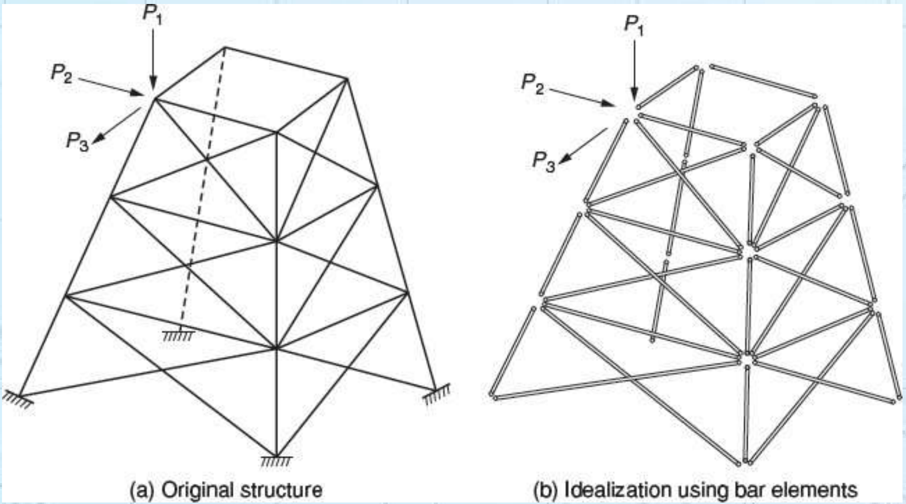
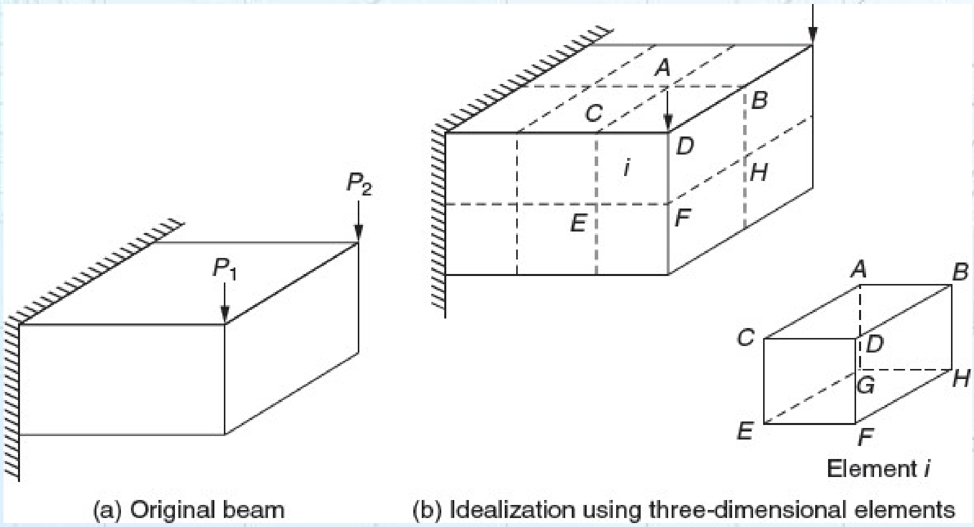


Fig. 5.17 A Truss Structure.

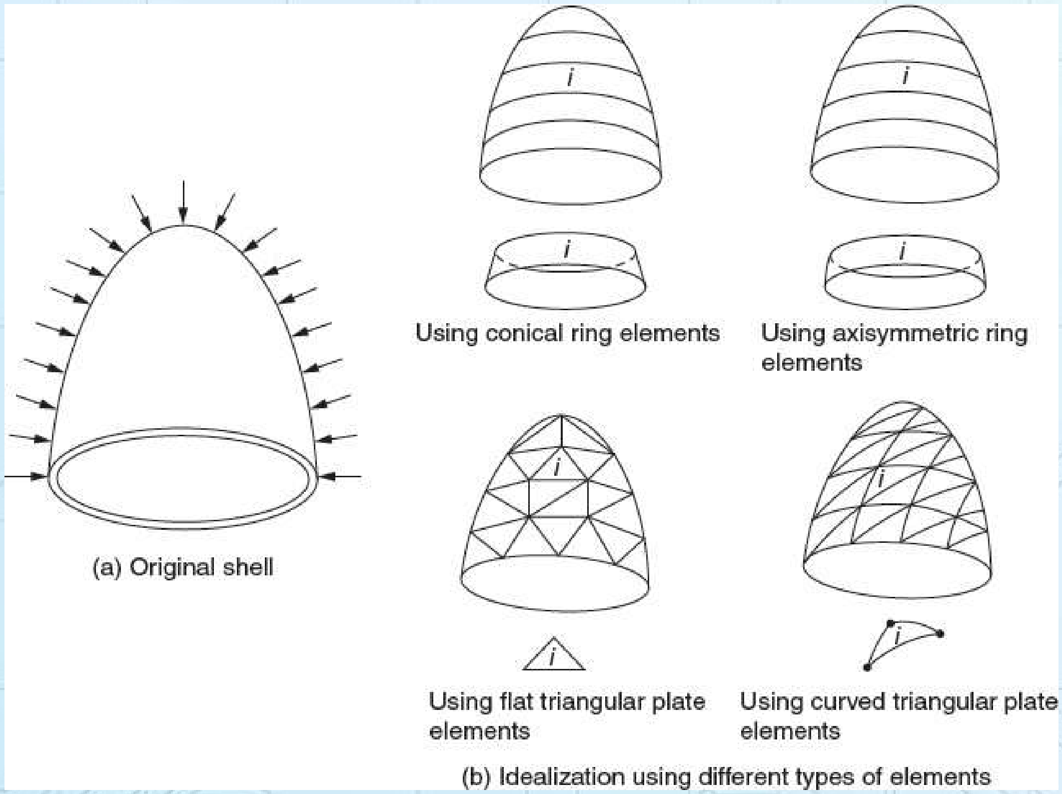
- In the case of stress analysis of the short beam shown in Fig.5.18(a), the finite element idealization can be done using three-dimensional solid elements as shown in Fig. 5.18(b).



(a) Original beam (b) Idealization using three-dimensional elements

Fig. 5.18 A Short Beam

- In some cases one has to choose the type of elements judicially. As an example, consider the problem of analysis of the thin walled shell shown in Fig. 5.19(a). In this case, the shell can be idealized by several types of elements as shown in Fig. 5.19(b).



(a) Original shell (b) Idealization using different types of elements

Fig. 5.19 A Thin-Walled Shell under pressure.

- In some cases, we may have to use two or more types of elements for idealization. An example of this would be the analysis of an aircraft wing.
- Since the wing consists of top and bottom covers, stiffening webs, and flanges, three types of elements—namely, triangular plate elements (for covers), rectangular shear panels (for webs), and frame elements (for flanges)—have been used in the idealization shown in Fig. 5.20.

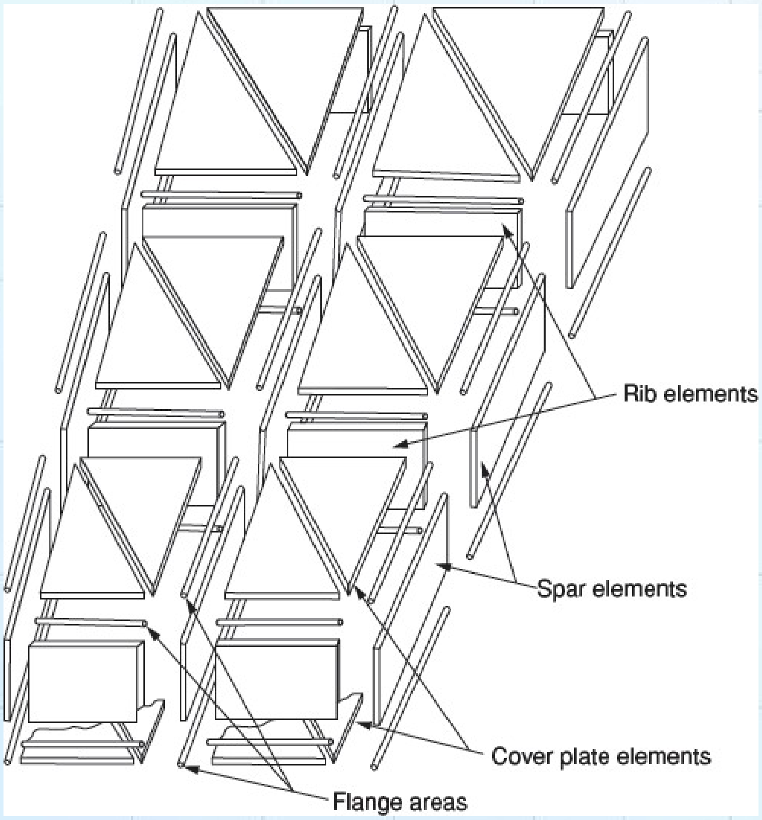
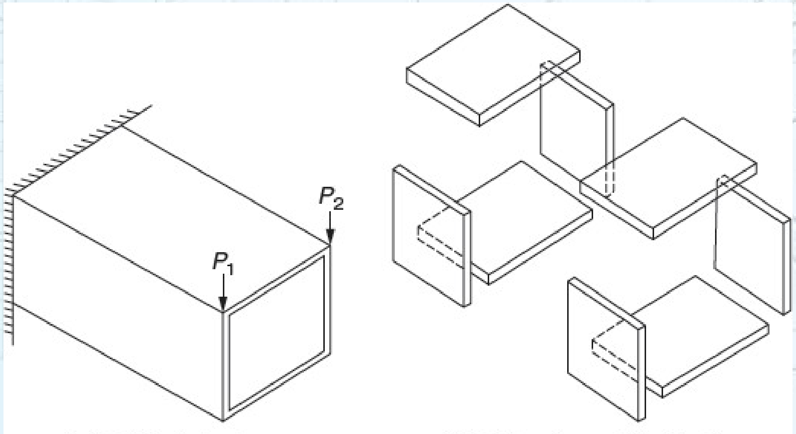


Fig. 5.20 Idealization of an Aircraft Wing Using Different Types of Elements.

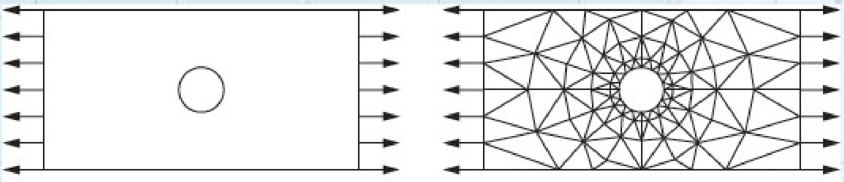
Step - 2: Size of Elements

- The size of elements influences the convergence of the solution directly, and hence it has to be chosen with care.
- If the size of the elements is small, the final solution is expected to be more accurate. However, we have to remember that the use of smaller-sized elements will also mean more computation time.
- Sometimes, we may have to use elements of different sizes in the same body. For example, in the case of stress analysis of the box beam shown in Fig. 5.21(a), the size of all the elements can be approximately the same, as shown in Fig. 5.21(b).
- However, in the case of stress analysis of a plate with a hole shown in Fig. 5.22 (a), elements of different sizes have to be used, as shown in Fig. 5.22 (b). The size of elements has to be very small near the hole (where stress concentration is expected) compared to distant places.



(a) Original structure (b) Finite element idealization

Fig. 5.21A Box Beam.

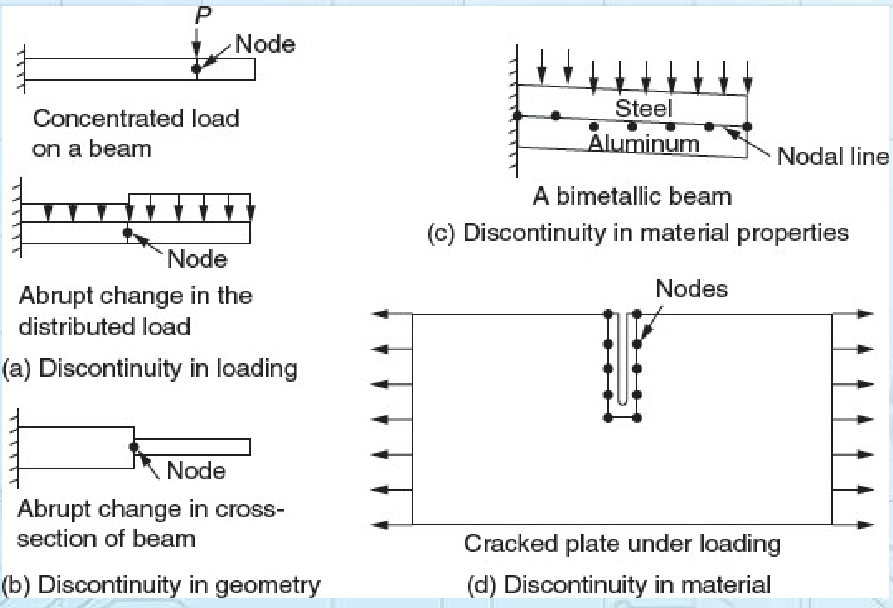


(a) Original structure (b) Idealization using elements of different sizes

Fig. 5.22 A Plate with a Hole.

- The aspect ratio describes the shape of the element in the assemblage of elements.
- For two-dimensional elements, the aspect ratio is taken as the ratio of the largest dimension of the element to the smallest dimension. Elements with an aspect ratio of nearly unity generally yield best results.

Step - 3: Location of Nodes



(a) Discontinuity in loading (b) Discontinuity in geometry (c) Discontinuity in material properties (d) Discontinuity in material

Fig. 5.23 Location of Nodes at Discontinuities.

- If the body has no abrupt changes in geometry, material properties, and external conditions (e.g., load and temperature), the body can be divided into equal subdivisions and hence the spacing of the nodes can be uniform.
- On the other hand, if there are any discontinuities in the problem, nodes have to be introduced at these discontinuities, as shown in Fig. 5.23.

Step - 4: Number of Elements

- The number of elements to be chosen for idealization is related to the accuracy desired, size of elements, and the number of dof involved.
- Although an increase in the number of elements generally means more accurate results, for any given problem, there will be a certain number of elements beyond which the accuracy cannot be significantly improved. This behavior is shown graphically in Fig. 5.24.
- Moreover, since the use of a large number of elements involves a large number of dof, we may not be able to store the resulting matrices in the available computer memory.

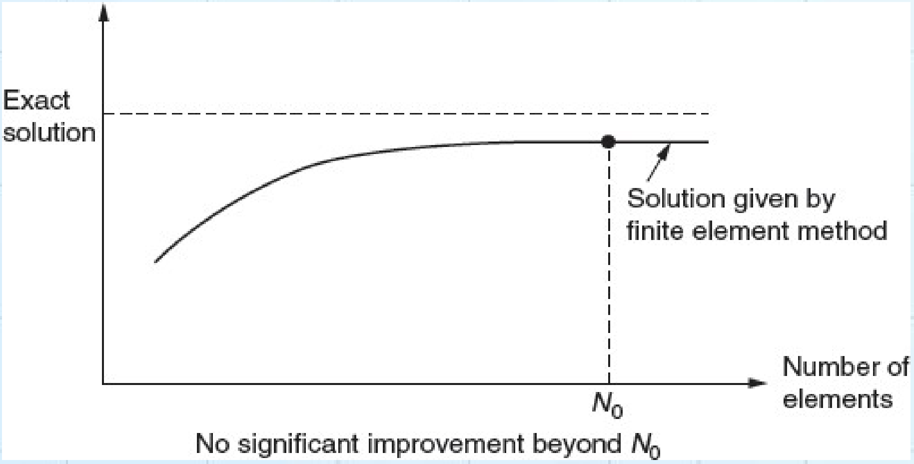
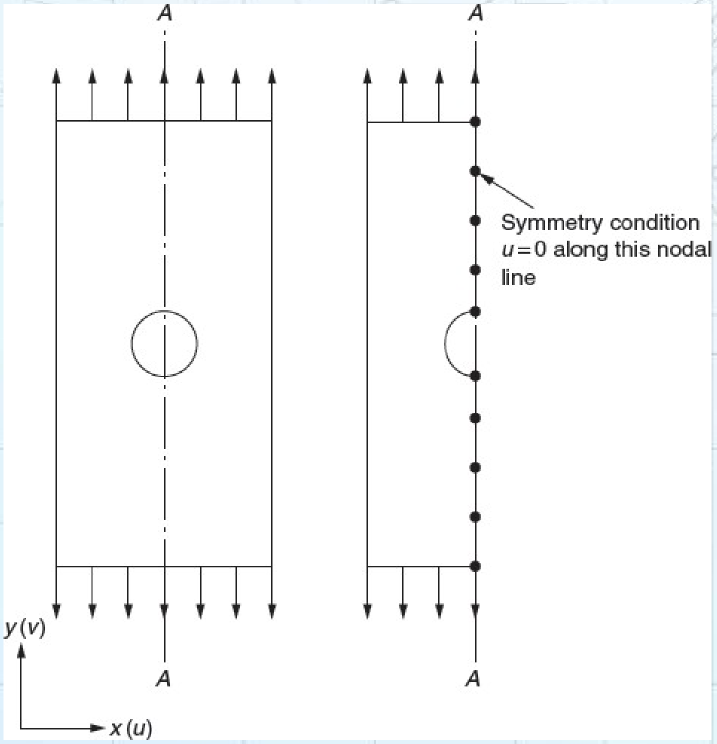


Fig. 5.24 Effect of Varying the Number of Elements.

Step - 5: Simplifications Afforded by the Physical Configuration of the Body

- If the configuration of the body as well as the external conditions are symmetric, we may consider only half of the body for finite element idealization. The symmetry conditions, however, have to be incorporated in the solution procedure.
- This is illustrated in Fig. 5.25, where only half of the plate with a hole, having symmetry in both geometry and loading, is considered for analysis. Since there cannot be a horizontal displacement along the line of symmetry AA, the condition that $u = 0$ has to be incorporated while finding the solution.



(a) Plate with hole (b) Only half of plate can be considered for analysis

Fig. 5.25A Plate with a Hole with Symmetric Geometry and Loading.

Step - 6: Finite Representation of Infinite Bodies

- In most of the problems, like in the analysis of beams, plates, and shells, the boundaries of the body or continuum are clearly defined. Hence, the entire body can be considered for element idealization.
- However, in some cases, as in the analysis of dams, foundations, and semi-infinite bodies, the boundaries are not clearly defined. In the case of dams (Figure 5.26), since the geometry is uniform and the loading does not change in the length direction, a unit slice of the dam can be considered for idealization and analyzed as a plane strain problem.

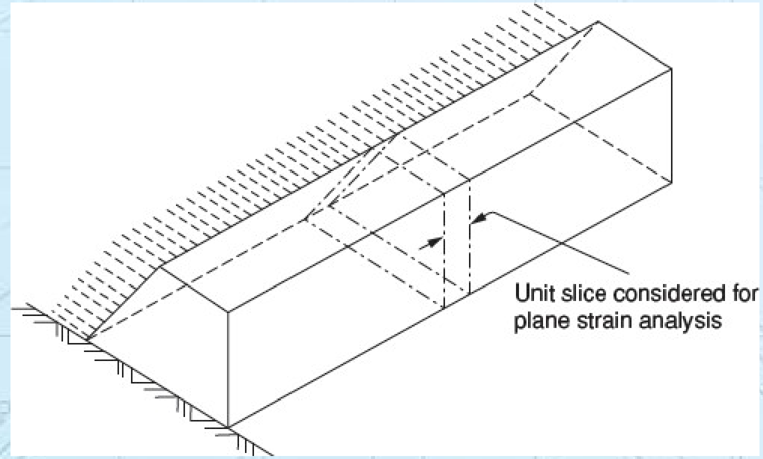
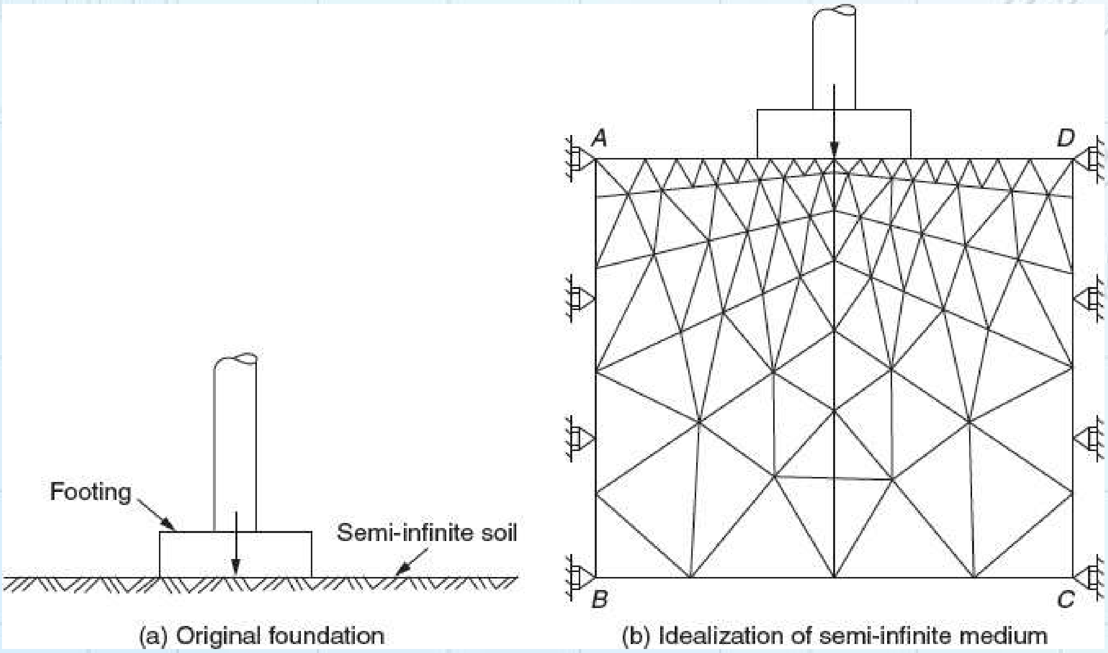


Fig. 5.26 A Dam with Uniform Geometry and Loading.

- However, in the case of the foundation problem shown in Fig. 5.27(a), we cannot idealize the complete semi-infinite soil by finite elements. Fortunately, it is not really necessary to idealize the infinite body.
- Since the effect of loading decreases gradually with increasing distance from the point of loading, we can consider only that much of the continuum in which the loading is expected to have a significant effect as shown in Fig. 5.27(b). Once the significant extent of the infinite body is identified as shown in Fig. 5.27(b), the boundary conditions for this finite body have to be incorporated in the solution.



(a) Original foundation

(b) Idealization of semi-infinite medium

Fig. 5.27 A Foundation under Concentrated Load.

5.11 Automatic Mesh Generation

- Mesh generation is the process of dividing a physical domain into smaller subdomains (called elements) to facilitate an approximate solution of the governing ordinary or partial differential equation.
- For this, one-dimensional domains (straight or curved lines) are subdivided into smaller line segments, two-dimensional domains (planes or surfaces) are subdivided into triangle or quadrilateral shapes, and three-dimensional domains (volumes) are subdivided into tetrahedron and hexahedron shapes.
- If the physical domain is simple and the number of elements used is small, mesh generation can be done manually. However, most practical problems, such as those encountered in aerospace, automobile, and construction industries have complex geometries that require the use of thousands and sometimes millions of elements.
- In such cases, the manual process of mesh generation is impossible and we have to use automatic mesh generation schemes based on the use of a CAD or solid modeling package.

- Automatic mesh generation involves the subdivision of a given domain, which may be in the form of a curve, surface, or solid (described by a CAD or solid modeling package) into a set of nodes (or vertices) and elements (subdomains) to represent the domain as closely as possible subject to the specified element shape and size restrictions.
- Many automatic mesh generation schemes use a “bottom-up” approach in that nodes (or vertices or corners of the domain) are meshed first, followed by curves (boundaries), then surfaces, and finally solids.
- Thus, for a given geometric domain of the problem, nodes are first placed at the corner points of the domain, and then nodes are distributed along the geometric curves that define the boundaries.
- Next, the boundary nodes are used to develop nodes in the surface(s), and finally the nodes on the various surfaces are used to develop nodes within the given volume (or domain).
- The nodes or mesh points are used to define line elements if the domain is one-dimensional, triangular, or quadrilateral elements if the domain is two-dimensional, and tetrahedral or hexahedral elements if the domain is three-dimensional.
- The automatic mesh generation schemes are usually tied to solid modeling and computer aided design schemes. When the user supplies information on the surfaces and volumes of the material domains that make up the object or system, an automatic mesh generator generates the nodes and elements in the object.
- The user can also specify minimum permissible element sizes for different regions of the object. Many mesh generation schemes first create all the nodes and then produce a mesh of triangles by connecting the nodes to form triangles (in a plane region).
- In a particular scheme, known as Delaunay triangulation, the triangular elements are generated by maximizing the sum of the smallest angles of the triangles; thus the procedure avoids generation of thin elements.

5.12Analysis of Trusses

- The links of a truss are two-force members, where the direction of loading is along the axis of the member. Every truss element is in direct tension or compression.
- All loads and reactions are applied only at the joints and all members are connected together at their ends by frictionless pin joints. This makes the truss members very similar to a 1D spar element.

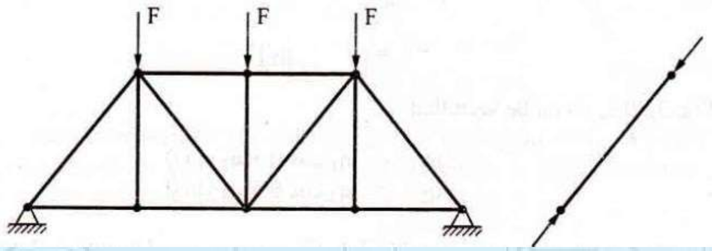
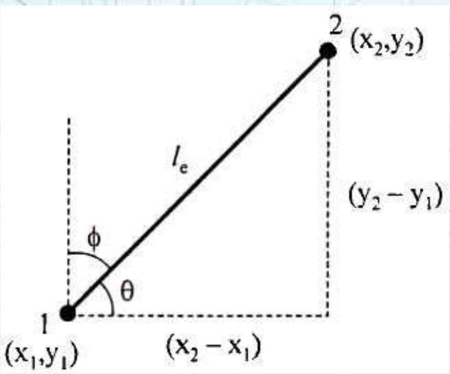


Fig. 5.28 Truss



The direction cosines *l* and *m* can be expressed as:

$$l = \cos\theta = \frac{x_2 - x_1}{l_e}$$

$$m = \cos\phi = \sin\theta = \frac{y_2 - y_1}{l_e}$$

$$l_e = \sqrt{(x_2 - x_1)^2 - (y_2 - y_1)^2}$$

$q_1' = q_1 l + q_2 m$

$q_2' = q_3 l + q_4 m$

Fig. 5.29 Direction Cosines

$$[K] = \frac{A_e E_e}{L_e}\begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix}$$

$$\sigma = \frac{E_e}{l_e}\begin{Bmatrix} -l & -m & l & m \end{Bmatrix}\{q\}$$

- **Thermal Effect In Truss Member**

(1) Thermal Load , P = $A_e E_e \alpha_e \Delta T \begin{Bmatrix} -l \\ -m \\ l \\ m \end{Bmatrix}$

(2) Stress for an element, $\sigma = \frac{E_e}{l_e}\begin{Bmatrix} -l & -m & l & m \end{Bmatrix}\{q\} - E_e \alpha \Delta t$

(3) Remaining steps will be same as earlier.

Example 5.6: A two member truss is as shown in Fig. 5.30. The cross-sectional area of each member is 200 mm^2 and the modulus of elasticity is 200 GPa. Determine the deflections, reactions and stresses in each of the members.

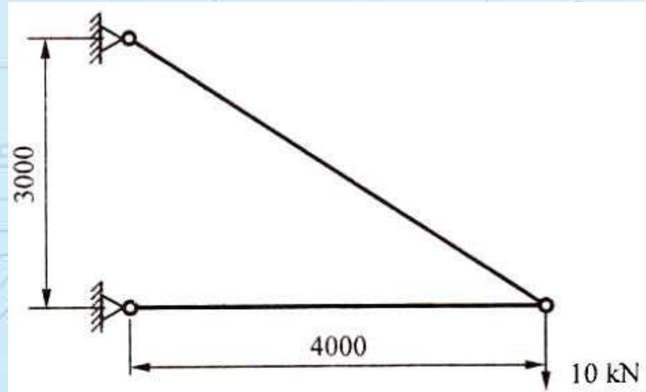


Fig. 5.30

In global terms, each node would have 2 dof. These dof are marked as shown in Fig.5.31. The position of the nodes, with respect to origin (considered at node l) are as tabulated below:

Node	Xi	Yi
1	0	0
2	4000	0
3	0	3000

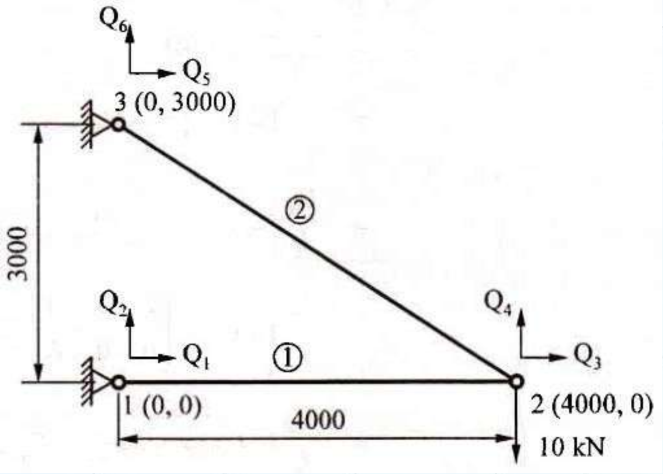


Fig.5.31

For all elements, A=200 mm^2
and E= 200 x 10^3 N/mm^2

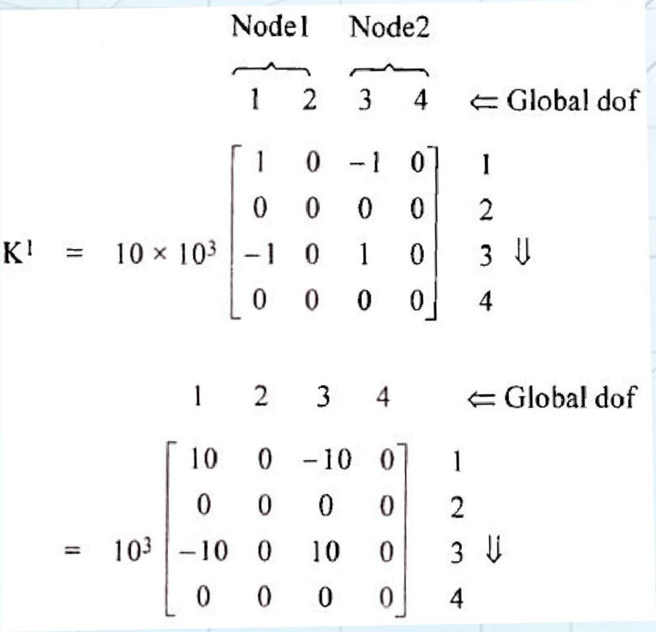
The element connectivity table with the relevant terms are:

Element	N$_i$	N$_j$	$l_e = \sqrt{(x_j - x_i)^2 + (y_j - y_i)^2}$	$\frac{A_e E_e}{l_e}$	$l = \frac{x_j - x_i}{l_e}$	$m = \frac{y_j - y_i}{l_e}$	l^2	m^2	lm
(1)	1	2	$\sqrt{(4000-0)^2 + (0-0)^2}$ $= 4000$	10000	$\frac{4000 - 0}{4000}$ $= 1$	$\frac{0-0}{4000} = 0$	1	0	0
(2)	2	3	$\sqrt{(0-4000)^2 + (3000-0)^2}$ $= 5000$	8000	$-\frac{4000}{5000}$ $= -0.8$	$\frac{3000}{5000} = 0.6$	0.64	0.36	-0.48

As each node has two dof in global form, for every element, the element stiffness matrix would be in a 4 x 4 form. For element 1 defined by nodes l-2,the dof are Q$_1$, Q$_2$, Q$_3$ andQ$_4$ and that for element 2 defined by nodes 2-3, would be Q$_3$, Q$_4$, Q$_5$ andQ$_6$. For a truss element

$$[K]^e = \frac{A_e E_e}{L_e}\begin{bmatrix} l^2 & lm & -l^2 & -lm \\ lm & m^2 & -lm & -m^2 \\ -l^2 & -lm & l^2 & lm \\ -lm & -m^2 & lm & m^2 \end{bmatrix}$$

Element 1: The element stiffness matrix would be :



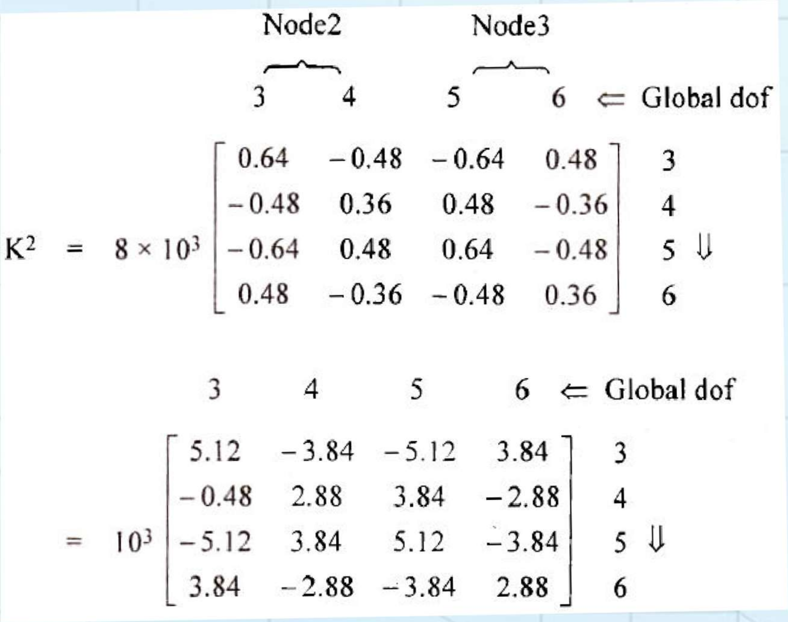
$$\begin{array}{ccl} \overbrace{1 \quad 2}^{\text{Node1}} & \overbrace{3 \quad 4}^{\text{Node2}} & \Leftarrow \text{Global dof} \end{array}$$

$$K^1 = 10 \times 10^3 \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{array}{l} 1 \\ 2 \\ 3 \;\Downarrow \\ 4 \end{array}$$

$$\begin{array}{ccccl} 1 & 2 & 3 & 4 & \Leftarrow \text{Global dof} \end{array}$$

$$= 10^3 \begin{bmatrix} 10 & 0 & -10 & 0 \\ 0 & 0 & 0 & 0 \\ -10 & 0 & 10 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{array}{l} 1 \\ 2 \\ 3 \;\Downarrow \\ 4 \end{array}$$

Element 2: The element stiffness matrix would be :



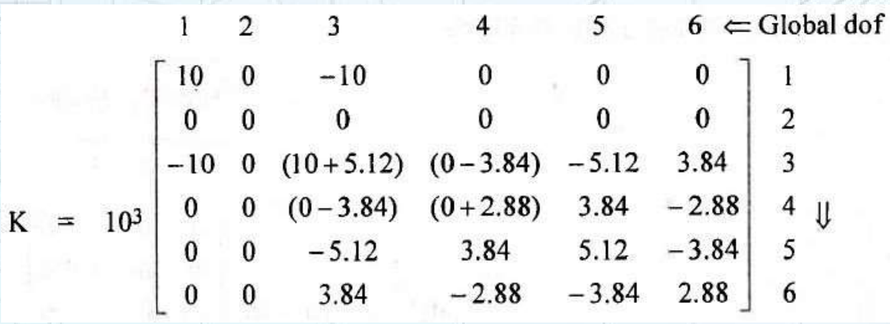
$$\begin{array}{ccl} \overbrace{3 \qquad 4}^{\text{Node2}} & \overbrace{5 \qquad 6}^{\text{Node3}} & \Leftarrow \text{Global dof} \end{array}$$

$$K^2 = 8 \times 10^3 \begin{bmatrix} 0.64 & -0.48 & -0.64 & 0.48 \\ -0.48 & 0.36 & 0.48 & -0.36 \\ -0.64 & 0.48 & 0.64 & -0.48 \\ 0.48 & -0.36 & -0.48 & 0.36 \end{bmatrix} \begin{array}{l} 3 \\ 4 \\ 5 \;\Downarrow \\ 6 \end{array}$$

$$\begin{array}{ccccl} 3 & 4 & 5 & 6 & \Leftarrow \text{Global dof} \end{array}$$

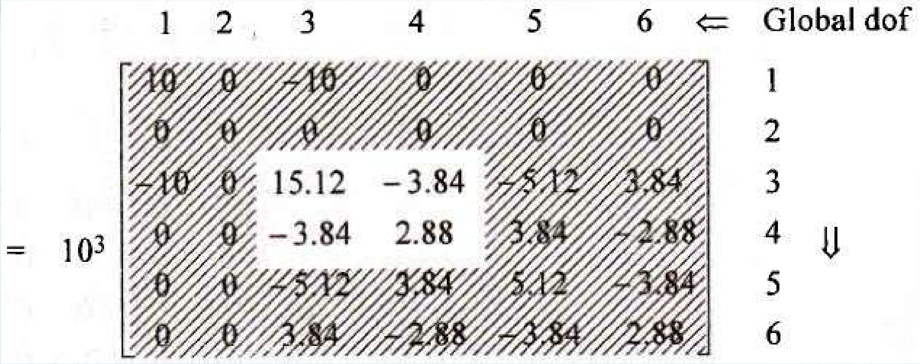
$$= 10^3 \begin{bmatrix} 5.12 & -3.84 & -5.12 & 3.84 \\ -0.48 & 2.88 & 3.84 & -2.88 \\ -5.12 & 3.84 & 5.12 & -3.84 \\ 3.84 & -2.88 & -3.84 & 2.88 \end{bmatrix} \begin{array}{l} 3 \\ 4 \\ 5 \;\Downarrow \\ 6 \end{array}$$

The global stiffness matrix would be:



$$K = 10^3 \begin{array}{c} \\ \left[\begin{array}{cccccc} 10 & 0 & -10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -10 & 0 & (10+5.12) & (0-3.84) & -5.12 & 3.84 \\ 0 & 0 & (0-3.84) & (0+2.88) & 3.84 & -2.88 \\ 0 & 0 & -5.12 & 3.84 & 5.12 & -3.84 \\ 0 & 0 & 3.84 & -2.88 & -3.84 & 2.88 \end{array}\right] \end{array} \begin{array}{c} \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{array} \Downarrow$$

Column global dof: 1, 2, 3, 4, 5, 6 $\Leftarrow$ Global dof



$$= 10^3 \begin{array}{ccccccc} 1 & 2 & 3 & 4 & 5 & 6 & \Leftarrow \text{Global dof} \\ 10 & 0 & -10 & 0 & 0 & 0 & 1 \\ 0 & 0 & 0 & 0 & 0 & 0 & 2 \\ -10 & 0 & 15.12 & -3.84 & -5.12 & 3.84 & 3 \\ 0 & 0 & -3.84 & 2.88 & 3.84 & -2.88 & 4 \\ 0 & 0 & -5.12 & 3.84 & 5.12 & -3.84 & 5 \\ 0 & 0 & 3.84 & -2.88 & -3.84 & 2.88 & 6 \end{array} \Downarrow$$

In this case, node 1 and node 3 are completely fixed and hence,

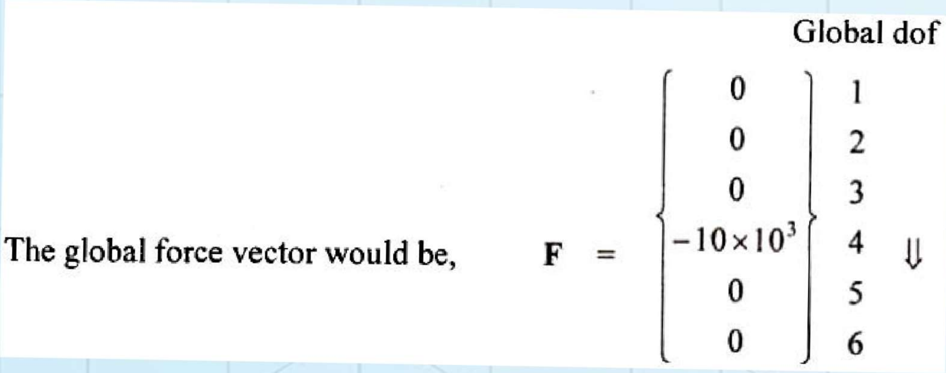
$$Q_1 = Q_2 = Q_5 = Q_6 = 0$$

Hence, rows and columns 1,2,5 and 6 can be eliminated

Also the external nodal forces,

$$F_1 = F_2 = F_3 = F_5 = F_6 = 0$$

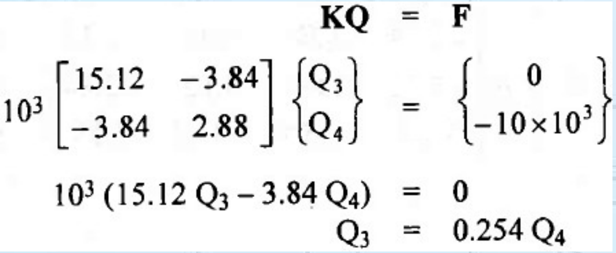
$$F4 = -10 \times 10^3 \text{ N}$$



The global force vector would be,

$$\mathbf{F} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ -10\times10^3 \\ 0 \\ 0 \end{Bmatrix} \begin{array}{c} \text{Global dof} \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \end{array} \Downarrow$$

In global form, after using the elimination approach



$$\mathbf{KQ} = \mathbf{F}$$

$$10^3 \begin{bmatrix} 15.12 & -3.84 \\ -3.84 & 2.88 \end{bmatrix} \begin{Bmatrix} Q_3 \\ Q_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10\times10^3 \end{Bmatrix}$$

$$10^3 (15.12\, Q_3 - 3.84\, Q_4) = 0$$

$$Q_3 = 0.254\, Q_4$$

$$10^3 (-3.84\, Q_3 + 2.88\, Q_4) = -10 \times 10^3$$

$- 3.84\, Q_3 + 2.88\, Q_4 = - 10$

$- 3.84\, (0.254\, Q_4) + 2.88\, Q_4 = -10$

$Q_4 = -5.25\text{mm}$

$Q_3 = -1.334\text{mm}$

The reactions can be found by using the equation:

$R = KQ - F$

$$\begin{Bmatrix} R_1 \\ R_2 \\ R_5 \\ R_6 \end{Bmatrix} = 10^3 \begin{bmatrix} 10 & 0 & -10 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & -5.12 & 3.84 & 5.12 & -3.84 \\ 0 & 0 & 3.84 & -2.88 & -3.84 & 2.88 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ -1.334 \\ -5.25 \\ 0 \\ 0 \end{Bmatrix} - \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix}$$

$R_1 = -10 \times 10^3 \times (-1.334) = 13340$ N

$R_2 = 0$ N

$R_5 = -5.12 \times 10^3 \times (-1.334) + 3.84 \times 10^3 \times (-5.25) = -13340$ N

$R_6 = 3.84 \times 10^3 \times (-1.334) - 2.88 \times 10^3 \times (-5.25) = 9997.44$ N

To determine stresses:□ □$\frac{E_e}{l_e}$ $-m \quad l \quad m$□q

□□l

Element 1:

$$\sigma_1 = \frac{200 \times 10^3}{4000} [-1 \quad 0 \quad 1 \quad 0] \begin{Bmatrix} 0 \\ 0 \\ -1.334 \\ -5.25 \end{Bmatrix}$$

$$= -66.7 \text{ N/mm}^2$$

Element 2:

$$\sigma_2 = \frac{200 \times 10^3}{5000} [0.8 \quad -0.6 \quad -0.8 \quad 0.6] \begin{Bmatrix} -1.334 \\ -5.25 \\ 0 \\ 0 \end{Bmatrix}$$

$$= 83.312 \text{ N/mm}^2$$

Example 5.7: Consider the four-bar truss shown in Fig. 5.32(a). It is given that $E = 29.5 \times 10^6$ psi and $A_e = 1\text{in.}^2$ for all elements. Complete the following:

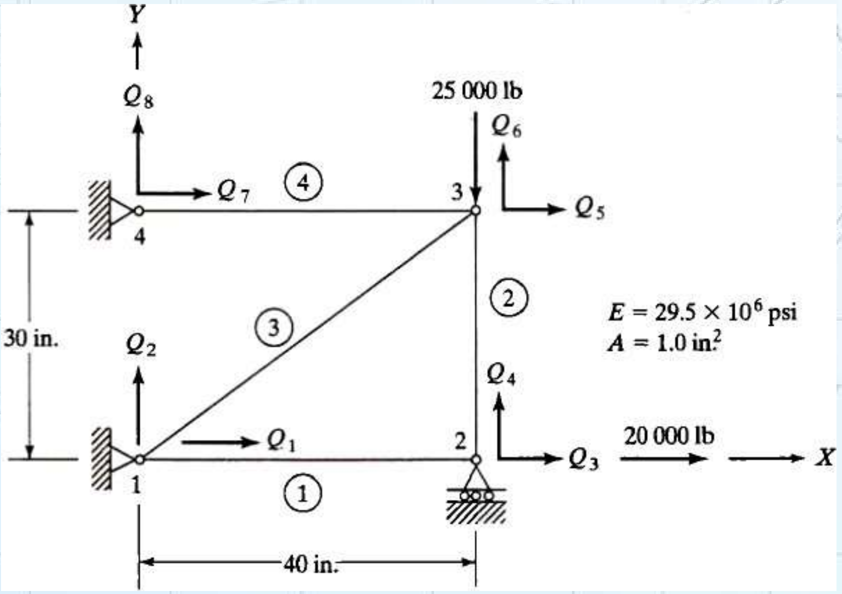
(a) Determine the element stiffness matrix for each element.

(b) Assemble the structural stiffness matrix K for the entire truss

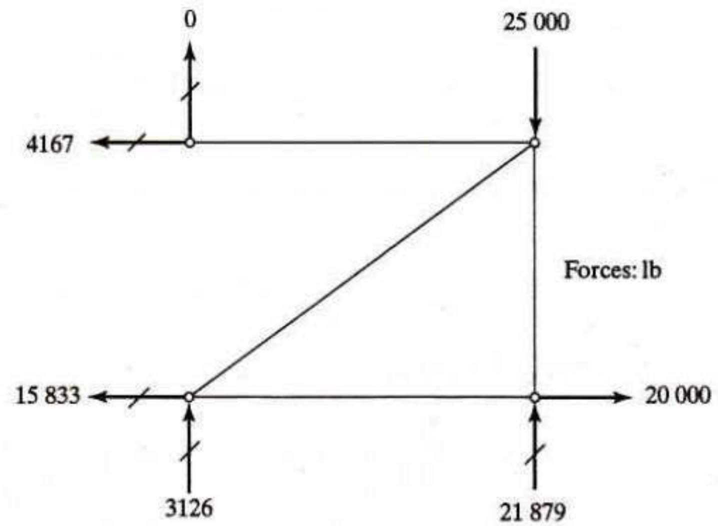
(c) Using the elimination approach, solve for the nodal displacement.

(d) Recover the stresses in each element.

(e) Calculate the reaction forces.



(a)



(b)

Fig. 5.32

(a) It is recommended that a tabular form be used for representing nodal coordinate data and element information. The nodal coordinate data are as follows:

Node	x	y
1	0	0
2	40	0
3	40	30
4	0	30

The element connectivity table is

Element	1	2
1	1	2
2	3	2
3	1	3
4	4	3

Note that the user has a choice in defining element connectivity. For example, the connectivity of element 2 can be defined as 2-3 instead of 3-2 as in the previous table. However, calculations of the direction cosines will be consistent with the adopted connectivity scheme. Using formulas, together with the nodal coordinate data and the given element connectivity information, we obtain the direction cosines table:

Element	l_e	l	m
1	40	1	0
2	30	0	-1
3	50	0.8	0.6
4	40	1	0

For example, the direction cosines of elements 3 are obtained as

$$l = (x_3 - x_1)/l_e = (40 - 0)/50 = 0.8 \text{ and } m = (y_3 - y_1)/l_e = (30 - 0)/50 = 0.6.$$

Now, the element stiffness matrices for element 1 can be written as

$$\mathbf{k}^1 = \frac{29.5 \times 10^6}{40} \begin{array}{c} \begin{array}{cccc} 1 & 2 & 3 & 4 \end{array} \leftarrow\!\downarrow \text{ Global dof} \\ \left[\begin{array}{cccc} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{array}\right] \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \end{array} \end{array}$$

The global dofs associated with element 1, which is connected between nodes 1 and2, are indicated in k^1 earlier. These global dofs are shown in Fig. 5.32(a) and assist in assembling the various element stiffness matrices. The element stiffness matrices of elements 2,3and 4 are as follows:

$$\mathbf{k}^2 = \frac{29.5 \times 10^6}{30} \begin{array}{c} \begin{array}{cccc} 5 & 6 & 3 & 4 \end{array} \\ \left[\begin{array}{cccc} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{array}\right] \begin{array}{c} 5 \\ 6 \\ 3 \\ 4 \end{array} \end{array}$$

$$\mathbf{k}^3 = \frac{29.5 \times 10^6}{50}\begin{array}{c} \begin{array}{cccc} 1 & 2 & 5 & 6 \end{array} \\ \left[\begin{array}{cccc} .64 & .48 & -.64 & -.48 \\ .48 & .36 & -.48 & -.36 \\ -.64 & -.48 & .64 & .48 \\ -.48 & -.36 & .48 & .36 \end{array}\right] \end{array} \begin{array}{c} \\ 1 \\ 2 \\ 5 \\ 6 \end{array}$$

$$\mathbf{k}^4 = \frac{29.5 \times 10^6}{40}\begin{array}{c} \begin{array}{cccc} 7 & 8 & 5 & 6 \end{array} \\ \left[\begin{array}{cccc} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{array}\right] \end{array} \begin{array}{c} \\ 7 \\ 8 \\ 5 \\ 6 \end{array}$$

(b) The structural stiffness matrix K is now assembled from the element stiffness matrices. By adding the element stiffness contributions, noting the element connectivity, we get

$$\mathbf{K} = \frac{29.5 \times 10^6}{600}\begin{array}{c} \begin{array}{cccccccc} 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 \end{array} \\ \left[\begin{array}{cccccccc} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ -15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\ -7.68 & -5.76 & 0 & 0 & 22.68 & 5.76 & -15.0 & 0 \\ -5.76 & -4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{array}\right] \end{array} \begin{array}{c} \\ 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \\ 7 \\ 8 \end{array}$$

(c) The structural stiffness matrix K given above needs to be modified to account for the boundary conditions. The elimination approach will be used here. The rows and columns corresponding to dofs 1, 2, 4, 7, and 8, which correspond to fixed supports, are deleted from the K matrix. The reduced finite element equations are given as

$$\frac{29.5 \times 10^6}{600}\begin{bmatrix} 15 & 0 & 0 \\ 0 & 22.68 & 5.76 \\ 0 & 5.76 & 24.32 \end{bmatrix}\begin{Bmatrix} Q_3 \\ Q_5 \\ Q_6 \end{Bmatrix} = \begin{Bmatrix} 20\,000 \\ 0 \\ -25\,000 \end{Bmatrix}$$

Solution of these equations yields the displacements

$$\begin{Bmatrix} Q_3 \\ Q_5 \\ Q_6 \end{Bmatrix} = \begin{Bmatrix} 27.12 \times 10^{-3} \\ 5.65 \times 10^{-3} \\ -22.25 \times 10^{-3} \end{Bmatrix} \text{in.}$$

The nodal displacement vector for the entire structure can therefore be written as

$Q = [0, 0, 27.12 \times 10^{-3}, 0, 5.65 \times 10^{-3}, -22.25 \times 10^{-3}, 0, 0]^T$ in.

(d) The stress in each element can now be determined as shown below.
The connectivity of element 1 is 1 - 2. Consequently, the nodal displacement vector for element l is given by $q = [0, 0, 27.72 \times 10^{-3}, 0]^T$

$$\sigma_1 = \frac{29.5 \times 10^6}{40}[-1 \quad 0 \quad 1 \quad 0]\begin{Bmatrix} 0 \\ 0 \\ 27.12 \times 10^{-3} \\ 0 \end{Bmatrix}$$

$$= 20\,000.0 \text{ psi}$$

The stress in member 2 is given by

$$\sigma_2 = \frac{29.5 \times 10^6}{30}[0 \quad 1 \quad 0 \quad -1]\begin{Bmatrix} 5.65 \times 10^{-3} \\ -22.25 \times 10^{-3} \\ -27.12 \times 10^{-3} \\ 0 \end{Bmatrix}$$

$$= -21\,880.0 \text{ psi}$$

Following similar steps, we get

σ_3 = 5208.0Psi

σ_4= 4L67.0Psi

(e) The final step is to determine the support reactions. We need to determine the reaction forces along dofs 1, 2, 4, 7and 8, which correspond to fixed supports. These are obtained by substituting for Q into the original finite element equation R = KQ - F. In this substitution, only those rows of K corresponding to the support dofs are needed, and F = 0 for those dofs. Thus, we have

$$\begin{Bmatrix} R_1 \\ R_2 \\ R_4 \\ R_7 \\ R_8 \end{Bmatrix} = \frac{29.5 \times 10^6}{600}\begin{bmatrix} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} 0 \\ 0 \\ 27.12 \times 10^{-3} \\ 0 \\ 5.65 \times 10^{-3} \\ -22.25 \times 10^{-3} \\ 0 \\ 0 \end{Bmatrix}$$

Which results in

$$\begin{Bmatrix} R_1 \\ R_2 \\ R_4 \\ R_7 \\ R_8 \end{Bmatrix} = \begin{Bmatrix} -15833.0 \\ 3126.0 \\ 21879.0 \\ -4167.0 \\ 0 \end{Bmatrix} \text{lb}$$

5.13 Principle of Minimum Potential Energy

- This principle states that for all kinematically admissible displacement fields corresponding to equilibrium extremize the total potential energy. If extreme condition is a minimum, the equilibrium is stable. It means that if the system is stable and steady, then total potential energy of the system is zero.

- The application of this principle can be made in a spring or elastic system subjected to the loading conditions.
- Consider a system of four springs as shown in Fig. 5.33subjected to the deflections under the application of force. This system has three nodal points.

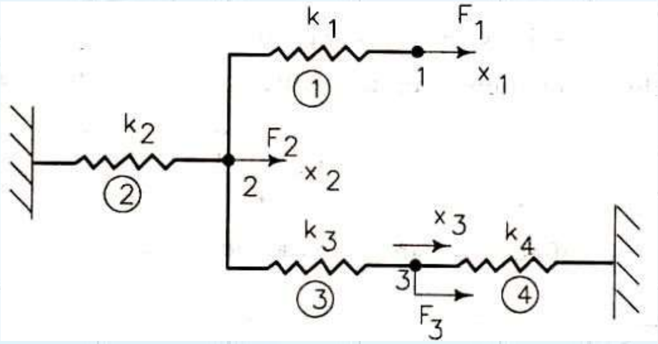


Fig. 5.33

Consider,a system of four spring having the nodes at point 1, 2, 3.

The potential energy P of the system is

$$P = \frac{1}{2}k_1\delta_1^2 + \frac{1}{2}k_2\delta_2^2 + \frac{1}{2}k_3\delta_3^2 + \frac{1}{2}k_4\delta_4^2 - F_1x_1 - F_3x_3$$

Where,

$\delta_1, \delta_2, \delta_3, \delta_4$ = deflections of the four springs

x_1, x_2, x_3 = displacement at nodal points 1, 2, 3

F_1, F_3 = forces at nodal points 1 and 3

k_1, k_2, k_3, k_4 = stiffnesses of four springs

$\delta_1 = x_1 - x_2$

$\delta_2 = x_2$

$\delta_3 = x_3 - x_2$

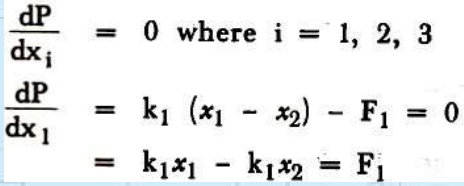
$\delta_4 = -x_3$

Potential energy of the system (P) is given by

$$P = \frac{1}{2}k_1(x_1 - x_2)^2 + \frac{1}{2}k_2x_2^2 + \frac{1}{2}k_3(x_3 - x_2)^2 + \frac{1}{2}k_4x_3^2 - F_1x_1 - F_3x_3$$

For equilibrium of this three degrees of freedom system, we need to minimize (P) with respect to x_1, x_2 and x_3.

According to principle of minimum potential energy, the potential energy is differentiated with respect to each displacement and equated to zero for minimum condition of potential energy.



$$\frac{dP}{dx_i} = 0 \text{ where } i = 1, 2, 3$$

$$\frac{dP}{dx_1} = k_1(x_1 - x_2) - F_1 = 0$$

$$= k_1x_1 - k_1x_2 = F_1$$

$$\frac{dP}{dx_2} = -k_1(x_1 - x_2) + k_2 x_2 - k_3(x_3 - x_2) = 0$$

$$= -k_1 x_1 + (k_1 + k_2 + k_3)x_2 - k_3 x_3 = 0$$

$$\frac{dP}{dx_3} = k_3(x_3 - x_2) + k_4 x_3 - F_3 = 0$$

$$= -k_3 x_2 + (k_3 + k_4)x_3 = F_3.$$

The above three equations can be written in matrix form as

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 + k_3 & -k_3 \\ 0 & -k_3 & k_3 + k_4 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ 0 \\ F_3 \end{bmatrix} \quad \text{.......(4)}$$

The x_1, x_2, x_3 deflections can be obtained by using numerical methods.

On the other hand, we proceed to write the equilibrium equations of the system by considering the equilibrium of each separate node as shown in Fig. 5.34.

$k_1\delta_1 = F_1$

$k_2\delta_2 - k_1\delta_1 - k_3\delta_3 = 0$

$k_3\delta_3 - k_4\delta_4 = F_3$

which is precisely the set of equations represented in Eq. (4).

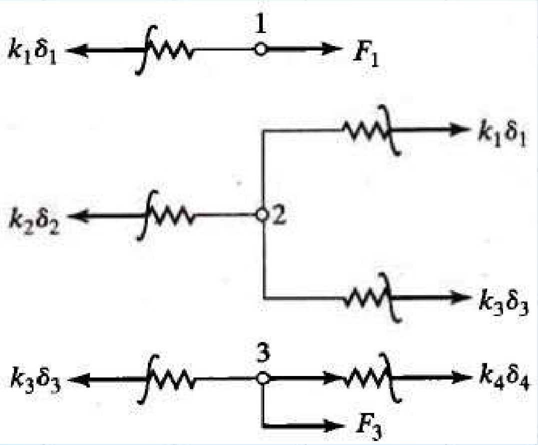


Fig. 5.34.

We see clearly that the set of equations (4) is obtained in a routine manner using the potential energy approach, without any reference to the free-body diagrams. This makes the potential energy approach attractive for large and complex problems.

Example 5.8: Calculate the deflections of the points at node 1, 2 and 3 for the spring system shown in Fig. 5.33. The stiffnesses are k_1 = 80 N/mm, k_2= 50 N/mm, k_3=60 N/mm, k_4 = 40 N/mm, the loads are F_1= 40 N, F_3 = 60 N.

Find the deflections x_1, x_2 and x_3.

The model of the above system is used and stiffness, deflection and load matrix can be written as

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 + k_3 & -k_3 \\ 0 & -k_3 & k_3 + k_4 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} F_1 \\ 0 \\ F_3 \end{bmatrix}$$

$$\begin{bmatrix} 80 & -80 & 0 \\ -80 & 80 + 50 + 60 & -60 \\ 0 & -60 & 60 + 40 \end{bmatrix} \begin{bmatrix} x_1 \\ x_2 \\ x_3 \end{bmatrix} = \begin{bmatrix} 40 \\ 0 \\ 60 \end{bmatrix}.$$

Solving the above matrices, the value of x_1= 0.4 mm, x_2= 0.1 mm, x_3= 0.66 mm.

Example 5.9: Fig. 5.35 shows a cluster of four springs. One end of the assembly is fixed and a force of 1000 N is applied at the end. Using the finite element method, determine:

(a) The deflection of each spring.
(b) The reaction forces at support.

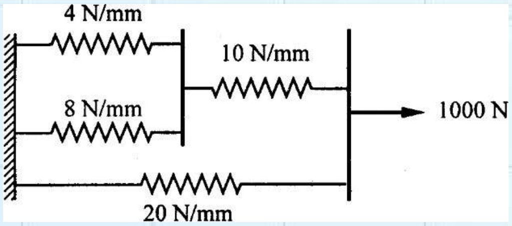


Fig. 5.35

The system of springs can be represented by a finite element model as shown in Fig. 5.36

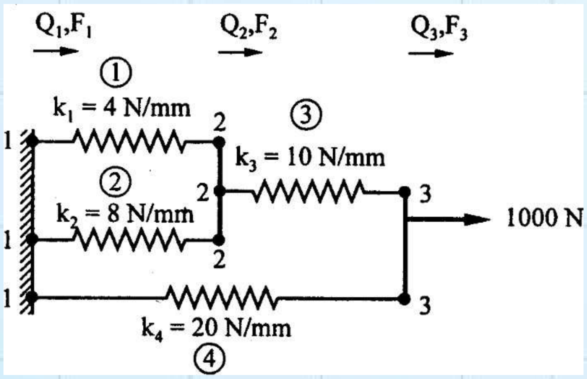
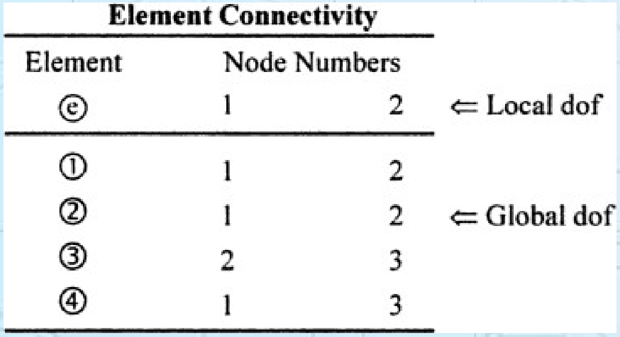


Fig. 5.36

The element connectivity table is as shown:



Element Connectivity

Element	Node Numbers		
ⓔ	1	2	⇐ Local dof
①	1	2	
②	1	2	⇐ Global dof
③	2	3	
④	1	3	

The element stiffness matrices are as under:

$$K^1 = \begin{array}{cc|l} 1 & 2 & \Leftarrow \text{ Global dof} \\ \left[\begin{matrix} 4 & -4 \\ -4 & 4 \end{matrix}\right. & \left.\begin{matrix} \\ \end{matrix}\right] & \begin{matrix} 1 \\ 2 \end{matrix} \quad \Downarrow \end{array}$$

$$K^2 = \begin{array}{cc|l} 1 & 2 & \Leftarrow \text{ Global dof} \\ \left[\begin{matrix} 8 & -8 \\ -8 & 8 \end{matrix}\right. & \left.\begin{matrix} \\ \end{matrix}\right] & \begin{matrix} 1 \\ 2 \end{matrix} \quad \Downarrow \end{array}$$

$$K^3 = \begin{array}{cc|l} 2 & 3 & \Leftarrow \text{ Global dof} \\ \left[\begin{matrix} 10 & -10 \\ -10 & 10 \end{matrix}\right. & \left.\begin{matrix} \\ \end{matrix}\right] & \begin{matrix} 2 \\ 3 \end{matrix} \quad \Downarrow \end{array}$$

$$K^4 = \begin{array}{cc|l} 1 & 3 & \Leftarrow \text{ Global dof} \\ \left[\begin{matrix} 20 & -20 \\ -20 & 20 \end{matrix}\right. & \left.\begin{matrix} \\ \end{matrix}\right] & \begin{matrix} 1 \\ 3 \end{matrix} \quad \Downarrow \end{array}$$

The overall stiffness matrix would be:

$$K = \begin{array}{ccc|l} 1 & 2 & 3 & \Leftarrow \text{ Global dof} \\ 4+8+20 & -4-8 & -20 & 1 \\ -4-8 & 4+8+10 & -10 & 2 \quad \Downarrow \\ -20 & -10 & 10+20 & 3 \end{array}$$

$$= \begin{array}{ccc|l} 1 & 2 & 3 & \Leftarrow \text{ Global dof} \\ 32 & -12 & -20 & 1 \\ -12 & 22 & -10 & 2 \quad \Downarrow \\ -20 & -10 & 30 & 3 \end{array}$$

In global terms,

$$\mathbf{KQ} = \mathbf{F}$$

$$\begin{bmatrix} 32 & -12 & -20 \\ -12 & 22 & -10 \\ -20 & -10 & 30 \end{bmatrix} \begin{Bmatrix} Q_1 \\ Q_2 \\ Q_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix}$$

The boundary conditions for this problem are:

$Q_1 = 0$

$F_1 = F_2 = 0$

$F_3 = 1000$ N

$$\begin{bmatrix} \cancel{32} & \cancel{-12} & \cancel{-20} \\ \cancel{-12} & 22 & -10 \\ \cancel{-20} & -10 & 30 \end{bmatrix} \begin{Bmatrix} \cancel{0} \\ Q_2 \\ Q_3 \end{Bmatrix} = \begin{Bmatrix} \cancel{0} \\ 0 \\ 1000 \end{Bmatrix}$$

By elimination approach:

$22\,Q_2 - 10\,Q_3 = 0$

$$Q_2 = 0.4545\ Q_3$$

$$-10\ Q_2 + 30\ Q_3 = 1000$$

$$-10\ (0.4545\ Q_3) + 30\ Q_3 = 1000$$

$$Q_3 = 39.286 \text{ mm}$$

$$Q_2 = 0.4545 \times 39.286 = 17.857 \text{ mm}$$

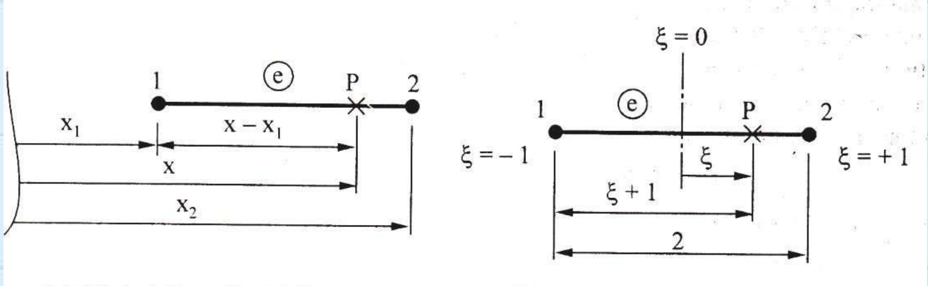
To determine the reaction forces,

$$R = KQ - F$$

$$\{R_1\} = [38 \quad -12 \quad -20] \begin{Bmatrix} 0 \\ 17.857 \\ 39.286 \end{Bmatrix} - \{0\}$$

$$\begin{aligned} R_1 &= -12 \times 17.857 - 20 \times 39.286 \\ &= -1000.004 \text{ N} \end{aligned}$$

5.14 Natural OR Intrinsic Coordinate System:



(a) Global Coordinate System (b) Natural Coordinate System

Coordinate Systems

Fig.5.37

- Consider a element (e), having node numbers 1 & 2 shown in Fig. 5.37 (a). The first node 1 would be at a distance x_1 and second node would be at a distance x_2 from the reference. A convenient coordinate system called as the natural coordinate system is defined, as it helps in formulating individual element matrix which can than be used to combine and form a global stiffness matrix.
- In natural coordinate system, the centre of the element is considered as 0 and the node 1 and node 2 are placed at a distance - 1 and + 1 respectively Fig. 5.37(b). The variable of measurement of the distance in this case is represented as ξ. Thus node 1 is at coordinate position $\xi = -1$ and node 2 is at $\xi = +1$. Total length of the element would thus be 2 units and this length of the element is covered in the range $\xi = -1$ to $+1$.
- To establish relationship between two coordinate system consider any point P situated at a distance x, in the Global coordinate system Fig.5.37 (a) and correspondingly at a distance ξ from the origin as shown in Fig.5.37 (b).

  Now

$$\frac{\text{Length of element in Natural system}}{\text{Length of element in Global system}} \square \frac{\text{Dist. of Point P from Node 1 in Natural system}}{\text{Dist. of Point P from Node 1 in Global system}}$$

$$\frac{2}{(x_2 - x_1)} = \frac{\xi + 1}{(x - x_1)}$$

$$\xi = \frac{2(x - x_1)}{(x_2 - x_1)} - 1 \qquad \underline{\qquad\qquad}(a)$$

Confirm the validity of equation (a)

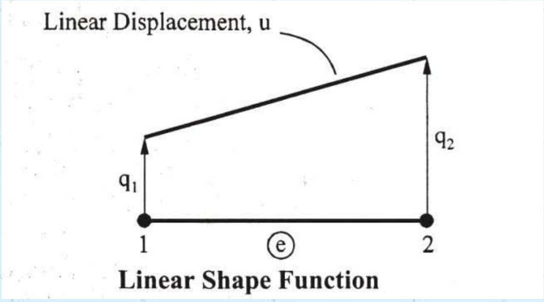
$$\text{At } x = x_1 \Rightarrow \xi = \frac{2(x_1 - x_1)}{(x_2 - x_1)} - 1 = 0 - 1 = -1$$

$$\text{At } x = x_2 \Rightarrow \xi = \frac{2(x_2 - x_1)}{(x_2 - x_1)} - 1 = 2 - 1 = 1$$

This confirm the relation of the two coordinate system.

5.15 Shape function in Natural Coordinate System:

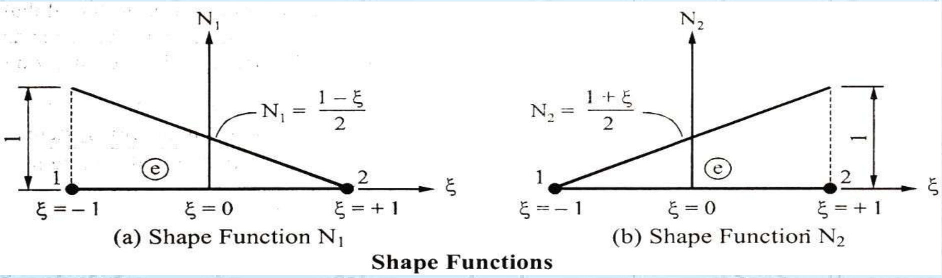
- The natural coordinate can be used to define shape functions. This makes it convenient to isolate the element from the continuum and develop the necessary element stiffness matrix. The shape function as defined earlier is used to interpolate the deflections or degree of freedom within the element. The accuracy of calculations would increase with increase in number of elements. Consider linear distribution as represented by Fig.5.38.



Linear Shape Function

Fig.5.38

- The shape function N_1 and N_2 in natural coordinate term can be developed by considering Fig.5.39 (a) and Fig.5.39 (b).



(a) Shape Function N_1

(b) Shape Function N_2

Shape Functions

Fig.5.39

- Fig. 5.39 (a) General line equation is

$y = mx + C$ $\left[m = \text{slope} = -\frac{1}{2} \right.$ $\left. \text{For } N_1 = 1,\ \xi = -1 \right]$

$\therefore N_1 = -\frac{1}{2}\xi + C$

$1 = -\frac{1}{2}(-1) + C$ $\quad \therefore C = \frac{1}{2}$

$\therefore N_1 = -\frac{1}{2}\xi + \frac{1}{2}$

$N_1(\xi) = \frac{1-\xi}{2}$

- Fig. 5.39(b) line equation is

$y = mx + C$ $\left[m = \text{slope} = \frac{1}{2} \right.$ $\left. \text{For } N_2 = 0,\ \xi = -1 \right]$

$\therefore N_2 = \frac{1}{2}\xi + C$

$0 = \frac{1}{2}(-1) + C$

$\therefore C = \frac{1}{2}$

$\therefore N_2 = \frac{1}{2}\xi + \frac{1}{2}$

$N_2(\xi) = \frac{1+\xi}{2}$

- Once the shape functions are defined, the linear displacement field within the element can be written in terms of nodal displacements q_1 and q_2 as….

$\therefore u = N_1 q_1 + N_2 q_2$ .... (b)

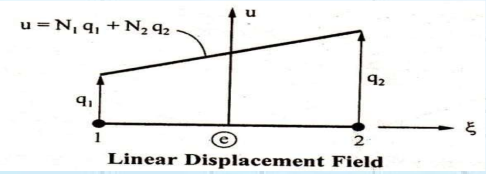


Fig.5.40

which in Matrix form will be

$$u = N\,q$$

where $N = [N_1 , N_2]$

$$q = [q_1 , q_2]^T = \begin{Bmatrix} q_1 \\ q_2 \end{Bmatrix}$$

- The term q is referred to as element displacement vector, and the verification of equation (b) can be done by considering the equation of shape functions.
- **At Node 1:** $\xi = -1$

$$N_1 = \frac{1-\xi}{2} = \frac{1-(-1)}{2} = 1$$

$$N_2 = \frac{1+\xi}{2} = \frac{1+(-1)}{2} = 0$$

So displacement at Node 1 will be

$$\begin{aligned} \therefore u &= N_1 q_1 + N_2 q_2 \\ &= 1\,(q_1) + 0\,(q_2) \\ &= q_1 \end{aligned}$$

- **At Node 2:** $\xi = +1$

$$N_1 = \frac{1-\xi}{2} = \frac{1-1}{2} = 0$$

$$N_2 = \frac{1+\xi}{2} = \frac{1+1}{2} = 1$$

So displacement at Node 2 will be

$$\begin{aligned} \therefore u &= N_2 q_2 + N_1 q_1 \\ &= 1\,(q_2) + 0\,(q_1) \\ &= q_2 \end{aligned}$$

- Thus it is seen that as per equation, the displacement at Node 1 and 2 are q_1 and q_2 which are the expected results.
- We know equation $\xi = \left[\frac{2\,(x - x_1)}{(x_2 - x_1)}\right] - 1$

$$\frac{\xi+1}{2}=\frac{x-x_1}{x_2-x_1}$$

$$\Rightarrow x=\left(\frac{\xi+1}{2}\right)(x_2-x_1)+x_1$$

$$=\left(\frac{\xi+1}{2}\right)x_2-\left(\frac{\xi+1}{2}\right)x_1+x_1$$

$$\Rightarrow x=\left(\frac{\xi+1}{2}\right)x_2+\left(-\frac{\xi+1}{2}+1\right)x_1$$

$$\Rightarrow x=\left(\frac{\xi+1}{2}\right)x_2+\left(\frac{1-\xi}{2}\right)x_1 \qquad \because \left(\frac{\xi+1}{2}\right)=N_2,\ \left(\frac{1-\xi}{2}\right)=N_1$$

$$\Rightarrow x=N_1x_1+N_2x_2 \qquad \left\{\begin{array}{l} x=N_1x_1+N_2x_2\\ u=N_1q_1+N_2q_2\end{array}\right\}$$

- Comparing equation, it is seen that both, the displacement u and the coordinate x can be interpolated within the element using the same shape function N_1 and N_2. This is referred to as the “Isoparametric Formulation”.

Example – 5.10: Temp. at Node 1 is 100° C and Node 2 is 40° C. The length of the element is 200 mm. Evaluate the shape function associated with Node 1 and Node 2. Calculate the temp. at point P situated at 150 mm from Node 1. Assume a linear shape function.

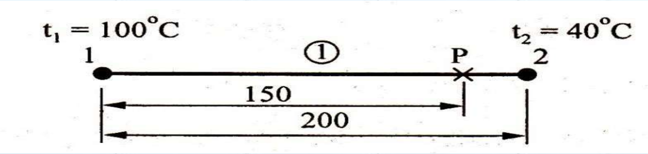


Fig. 5.41

Solution:

At Node 1:

$x_1 = 0$

$x_2 = 200$ mm

$x = 0$

$$\xi=\frac{2(x-x_1)}{(x_2-x_1)}-1=-1$$

$$N_1=\left(\frac{1-\xi}{2}\right)=1$$

$$N_2=\left(\frac{1+\xi}{2}\right)=0$$

At Node 2:

$x_1 = 0$

$x_2 = 200$ mm

$x = 200$ mm

$$\xi = \frac{2(x - x_1)}{(x_2 - x_1)} - 1 = \frac{2(200 - 0)}{(200 - 0)} - 1 = 1$$

$$N_1(\xi) = \frac{1 - \xi}{2} = 0$$

$$N_2(\xi) = \frac{1 + \xi}{2} = 1$$

At Point P:

$x_1 = 0 \Rightarrow t_1 = 100^\circ C$

$x_2 = 200$ mm $\Rightarrow t_2 = 40^\circ C$

$x = 150$ mm

$$\xi = \frac{2(x - x_1)}{(x_2 - x_1)} - 1 = \frac{2(150 - 0)}{(200 - 0)} - 1 = 0.5$$

$$N_1(\xi) = \frac{1 - \xi}{2} = \frac{1 - 0.5}{2} = 0.25$$

$$N_2(\xi) = \frac{1 + \xi}{2} = \frac{1 + 0.5}{2} = 0.75$$

$$\therefore t = N_1 t_1 + N_2 t_2$$

$$= 0.25(100) + 0.75(40) \therefore t = 55$$

Example – 5.11 : A 1D spar element having a linear shape function as shown in fig. If the temp.at Node 1 is 50° C and Node 2 is -20°C. Find the temp. at point P.

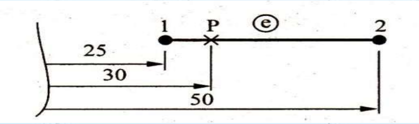


Fig. 5.42

Solution:

At Point P:

$x_1 = 25 \Rightarrow t_1 = 50^\circ C$

$x_2 = 50 \Rightarrow t_2 = -20^\circ C$

$x = 30$

Let

$$\xi = \frac{2(x - x_1)}{(x_2 - x_1)} - 1 = \frac{2(30 - 25)}{(50 - 25)} - 1 = -0.6$$

Now

$$N_1 = \frac{1-\xi}{2} = \frac{1-(-0.6)}{2} = 0.8$$

$$N_2 = \frac{1+\xi}{2} = \frac{1+(-0.6)}{2} = 0.2$$

$$t = N_1 t_1 + N_2 t_2$$
$$= 0.8(50) + 0.2(-20)$$
$$t = 30^\circ C$$

Example – 5.12: Consider an element having a linear shape function shown in fig. Evaluate the natural coordinate and shape functions for point P. If the displacement at Node 1 and Node 2 are 2 mm and -1 mm respectively, determine the value of displacement at point P. Also determine in global terms the point where the displacement would be zero. Also determine the shape function at zero displacement point.

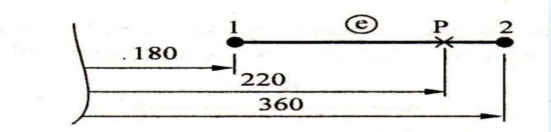


Fig. 5.43

Solution:

At point P:

$x_1 = 180$ mm $\quad q_1 = 2$ mm

$x_2 = 360$ mm $\quad q_2 = -1$ mm

$x = 220$ mm

Let

$$\xi = \frac{2(x - x_1)}{(x_2 - x_1)} - 1 = \frac{2(220 - 180)}{(360 - 180)} - 1 = -0.556$$

$$N_1 = \frac{1-\xi}{2} = \frac{1-(-0.556)}{2} = 0.778$$

$$N_2 = \frac{1+\xi}{2} = \frac{1+(-0.556)}{2} = 0.222$$

Let

$$u = N_1 q_1 + N_2 q_2$$
$$= 0.778(2) + 0.222(-1)$$
$$u = 1.334 \text{ mm}$$

To determine the position of point where displacement is zero ($q_1 = 2$ mm , $q_2 = -1$ mm).

Let

$$u = N_1 q_1 + N_2 q_2$$
$$0 = N_1(2) + N_2(-1)$$
$$\Rightarrow 2N_1 = N_2$$
$$\Rightarrow 2\left(\frac{1-\xi}{2}\right) = \left(\frac{1+\xi}{2}\right)$$
$$\Rightarrow 2 - 2\xi = 1 + \xi$$
$$\Rightarrow \xi = \frac{1}{3} = 0.333$$

Let

$$\xi = \frac{2(x - x_1)}{(x_2 - x_1)} - 1$$
$$0.333 = \frac{2(x - 180)}{(360 - 180)} - 1$$
$$x = 300 \text{ mm}$$
$$N_1 = \frac{1-\xi}{2} = \frac{1 - 0.333}{2} = 0.333$$
$$N_2 = 2N_1 = 2(0.333) = 0.667$$

– x can be find out

$$x = N_1 x_1 + N_2 x_2$$
$$= 0.333(180) + 0.667(360)$$
$$x = 300 \text{ mm}$$

Example – 5.13: Determine the temperature at $x = 40$ mm if the temperature at nodes $Ø_i = 120°$ C and $Ø_j = 80°$ C and $x_i = 10$ mm, $x_j = 60$ mm.

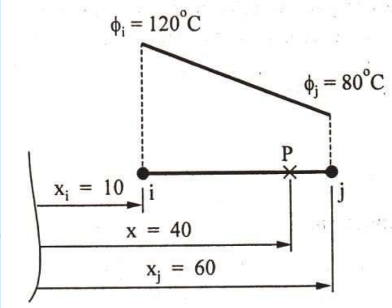


Fig. 5.44

Solution:

$x_i = 10$

$x_j = 60$

$x = 40$

$\theta_i = 120^\circ C$

$\theta_j = 80^\circ C$

Let

$$\xi = \frac{2(x - x_1)}{(x_2 - x_1)} - 1$$

$$= \frac{2(40 - 10)}{(60 - 10)} - 1$$

$$= 0.2$$

$$N_1(\xi) = \frac{1 - \xi}{2} = \frac{1 - 0.2}{2} = 0.4$$

$$N_2(\xi) = \frac{1 + \xi}{2} = \frac{1 + 0.2}{2} = 0.6$$

Let

$$\theta = N_1 \theta_i + N_2 \theta_j$$

$$= 0.4(120) + 0.6(80)$$

$$= 96^\circ C$$

References:

- Finite Element Method – S.S. Rao
- Introduction to Finite Elements in Engineering - Tirupathi R. Chandrupatla
- CAD/CAM & Automation – Farazdak Haideri
- CAD/CAM – Khandare