UNIT –II
ONE DIMENSIONAL & TWO DIMENSIONAL ELEMENTS

COORDINATES AND SHAPE FUNCTIONS

Consider a typical finite element e in Fig. 3.5a. In the local number scheme, the first node will be numbered 1 and the second node 2. The notation $x_1 = x$-coordinate of node 1, $x_2 = x$-coordinate of node 2 is used. We define a **natural** or **intrinsic** coordinate system, denoted by ξ, as

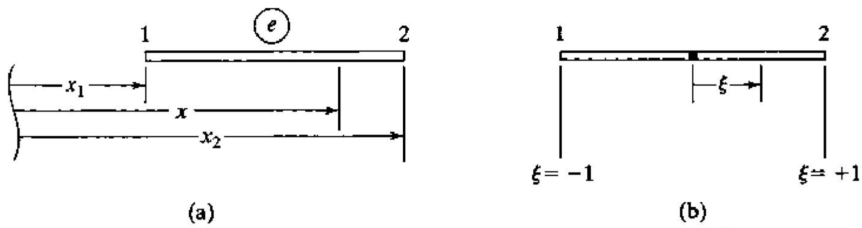


FIGURE 3.5 Typical element in x- and ξ-coordinates.

$$\xi = \frac{2}{x_2 - x_1}(x - x_1) - 1 \tag{3.4}$$

From Fig. 3.5b, we see that $\xi = -1$ at node 1 and $\xi = 1$ at node 2. The length of an element is covered when ξ changes from -1 to 1. We use this system of coordinates in defining shape functions, which are used in interpolating the displacement field.

Now *the unknown displacement field within an element will be interpolated by a linear distribution* (Fig. 3.6). This approximation becomes increasingly accurate as more elements are considered in the model. To implement this linear interpolation, linear **shape functions** will be introduced as

$$N_1(\xi) = \frac{1 - \xi}{2} \tag{3.5}$$

$$N_2(\xi) = \frac{1 + \xi}{2} \tag{3.6}$$

The shape functions N_1 and N_2 are shown in Figs. 3.7a and b, respectively. The graph of the shape function N_1 in Fig. 3.7a is obtained from Eq. 3.5 by noting that $N_1 = 1$ at $\xi = -1$, $N_1 = 0$ at $\xi = 1$, and N_1 is a straight line between the two points. Similarly, the graph of N_2 in Fig. 3.7b is obtained from Eq. 3.6. Once the shape functions are defined, the linear displacement field within the element can be written in terms of the nodal displacements q_1 and q_2 as

$$u = N_1 q_1 + N_2 q_2 \tag{3.7a}$$

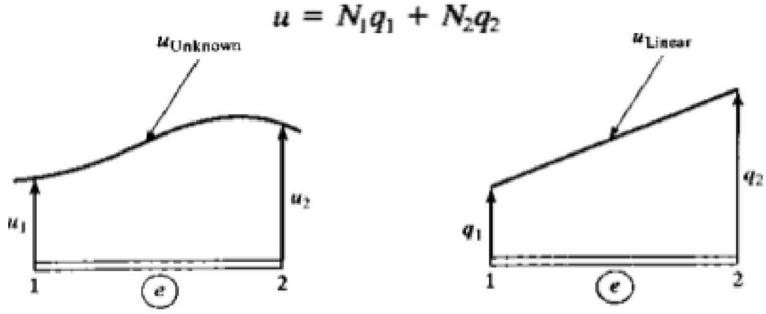


FIGURE 3.6 Linear interpolation of the displacement field within an element.

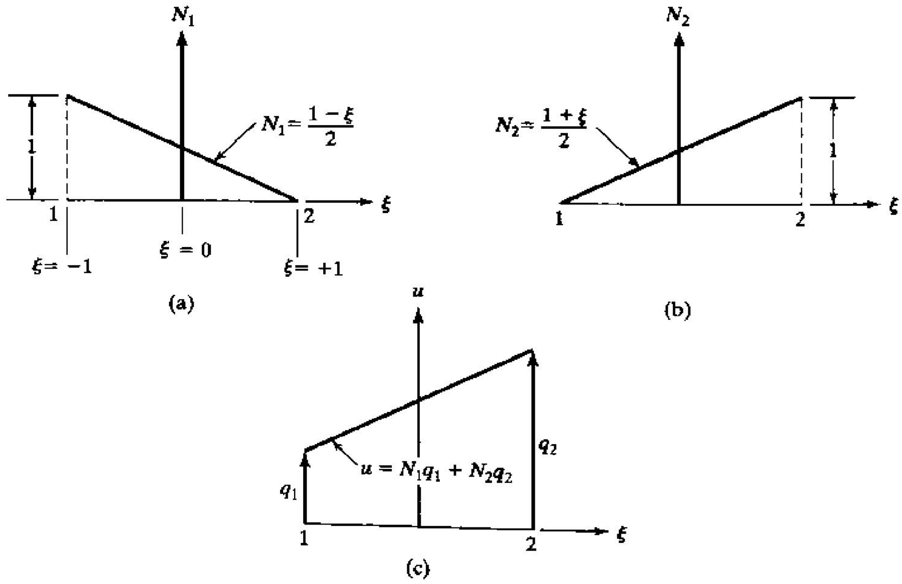


FIGURE 3.7 (a) Shape function N_1, (b) shape function N_2, and (c) linear interpolation using N_1 and N_2.

or, in matrix notation, as

$$u = \mathbf{Nq} \tag{3.7b}$$

where

$$\mathbf{N} = [N_1, N_2] \quad \text{and} \quad \mathbf{q} = [q_1, q_2]^T \tag{3.8}$$

In these equations, $\mathbf{q}$ is referred to as the *element displacement vector*. It is readily verified from Eq. 3.7a that $u = q_1$ at node 1, $u = q_2$ at node 2, and that u varies linearly (Fig. 3.7c).

It may be noted that the transformation from x to ξ in Eq. 3.4 can be written in terms of N_1 and N_2 as

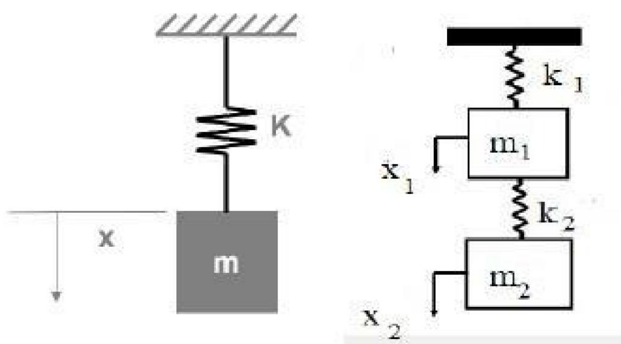
$$x = N_1x_1 + N_2x_2 \tag{3.9}$$

Comparing Eqs. 3.7a and 3.9, we see that both the displacement u and the coordinate x are interpolated within the element using the *same* shape functions N_1 and N_2. This is referred to as the *isoparametric* formulation in the literature.

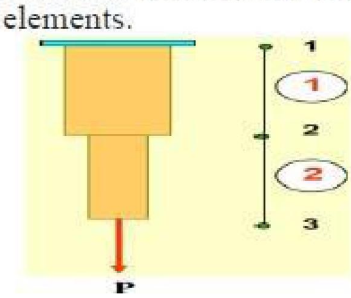
Though linear shape functions have been used previously, other choices are possible. Quadratic shape functions are discussed in Section 3.9. In general, shape functions need to satisfy the following:

Degrees of Freedom in FEA:

- Degree of Freedom (DoF) is a “possibility” to move in a defined direction. There are 6 DoF in a 3D space: you can move or rotate along axis x, y or z. Together, those components describe a motion in 3D. DoF in FEA also do other things: they control supports, information about stresses and more.
- Degree of freedom or DOF means the number of independent coordinates a structure can move. There are 6 DOF possible for a structure. They are movement on x,y and z axis and rotation about these axis.
- Whatever be the field, degree of freedom, dof in short, represents the minimum number of independent coordinates required to specify the position of every mass in the system uniquely.
- eg. A simple spring mass system as shown in Fig. which is constrained to move only in the vertical direction requires the displacement x only to specify the position of the mass m. Hence it has one degree of freedom.
- If we attach another spring and another mass below the first mass then each mass will undergo different displacement and hence we need to specify x1 and x2 which are the displacements of masses 1 and 2 respectively. Hence this has 2 dof.

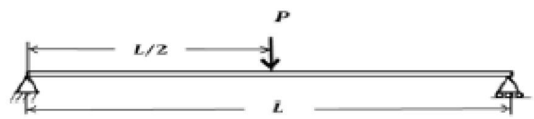


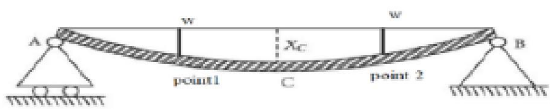
- Now coming to FEM when we want to find the stresses in a member subjected to axial loads such as the stepped bar shown below, since the bar is long and thin we can assume that the longitudinal displacements are significantly higher than the lateral displacements. So neglecting this lateral displacement we can discretise this system into two 2 noded elements.



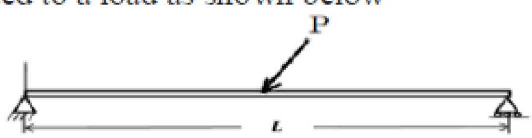
Finite Element Methods

- When we use a 2 noded element it is assumed that there is only on degree of freedom at each node namely the axial displacement of that node. So total degrees of freedom in this case for one element is 2.
- On the other hand in a beam element that is subjected to only vertical transverse loads, we require minimum 2 dofs. Why is this so?
- Take the case of a simply supported beam subjected to a central point load as shown in the figure below.





- If we specify the position of every point in the beam with only one variable namely the transverse displacement w, then if we look for two symmetrically placed points along the beam such as points 1 and 2, the displacements will be the same and equal to w. So if have to specifically refer to only one point uniquely we need one more variable that can be used to identify that point. Hence we introduce another variable namely the slope of the deflection curve.
- So a simple beam subjected to only vertical loads can be modelled using a beam element that has 2 dofs per node namely So total dof s for one two noded beam element is 2 x 2 =4.
- If the beam is subjected to a load as shown below



- Then there is an axial displacement that comes into the picture additionally. So we have to introduce one more dof namely axial displacement u at each node thus bringing the dof per node to 3 and total dof to 6.
- Similarly a 3 noded triangular element used to model a thin rectangular fin has one dof (variable) per node namely temperature so total dof is 3 x 1 =3. In a structural application there will be two dof per node namely u and v displacements. Hence total dof for a 3 noded triangular element for stress analysis will be 3 x 2 =6.
- A 4 noded tetrahedral solid element has 3 dof per node (u,v,and v displacements) when used in structural applications so total dof is 4 x 3 =12.
- So we need to understand the physical behaviour of the system and model it appropriately.

Shape Functions:

In the finite element analysis aim is to find the field variables at nodal points by rigorous analysis, assuming at any point inside the element basic variable is a function of values at nodal points of the element. This function which relates the field variable at any point within the element to the field variables of nodal points is called shape function. This is also called as interpolation function and approximating function. In two dimensional stress analysis in which basic field variable is displacement,

Shape functions are the polynomials meant to describe the variation of primary variable along the domain of element.

$$u = \Sigma N_i u_i, \; v = \Sigma N_i v_i \qquad \ldots(5.1)$$

where summation is over the number of nodes of the element. For example for three noded triangular element, displacement at $P(x, y)$ is

$$u = \Sigma N_i u_i = N_1 u_1 + N_2 u_2 + N_3 u_3$$

$$v = \Sigma N_i v_i = N_1 v_1 + N_2 v_2 + N_3 v_3$$

i.e.,

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N_1 & 0 & N_2 & 0 & N_3 & 0 \\ 0 & N_1 & 0 & N_2 & 0 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}$$

$$\begin{Bmatrix} u \\ v \end{Bmatrix} = \begin{bmatrix} N & 0 \\ 0 & N \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ v_1 \\ v_2 \\ v_3 \end{Bmatrix} = \begin{bmatrix} N_1 & N_2 & N_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & N_1 & N_2 & N_3 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ v_1 \\ v_2 \\ v_3 \end{Bmatrix}$$

$$\underset{2\times 1}{\{\delta\}} = \underset{2\times 6}{[N]} \underset{6\times 1}{\{\delta\}_e}$$

where q is displacement at any point in the element

$[N]$ shape function

$\{\delta\}_e$ is vector of nodal displacements

Similarly in case of 6 noded triangular element

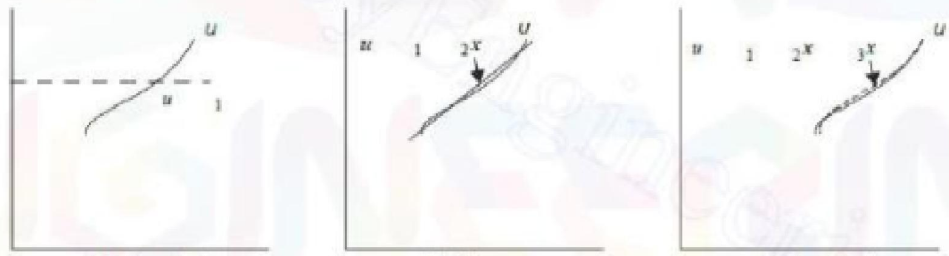
$$\underset{2\times 1}{\{\delta\}} = \underset{2\times 12}{[N]} \underset{12\times 1}{\{\delta\}_e}$$

POLYNOMIAL SHAPE FUNCTIONS

Polynomials are commonly used as shape functions. There are two reasons for using them:

(i) They are easy to handle mathematically i.e. differentiation and integration of polynomials is easy.

(ii) Using polynomial any function can be approximated reasonably well. If a function is highly nonlinear we may have to approximate with higher order polynomial. Fig. 5.1 shows approximation of a nonlinear one dimensional function by polynomials of different order.



Approximation with polynomials

One Dimensional Polynomial Shape Function

A general one dimensional polynomial shape function of nth Order is given by,

$$u(x) = \alpha_1 + \alpha_2 x + \alpha_3 x^2 + \dots \alpha_{n+1} x^n$$

In matrix form $u = [G]\{\alpha\}$

where $$[G] = [1, x, x^2 \dots x^n]$$

and $$\{\alpha\}^T = [\alpha_1\ \alpha_2\ \alpha_3 \dots \alpha_{n+1}]$$

Thus in one dimensional n^{th} order complete polynomial there are $m = n + 1$ terms.

Two Dimensional Polynomial Shape Function

A general form of two dimensional polynomial model is

$$u(x, y) = \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 x^2 + \alpha_5 xy + \alpha_6 y^2 + \alpha_7 x^3 \dots + \alpha_m y^n$$
$$v(x, y) = \alpha_{m+1} + \alpha_{m+2} x + \alpha_{m+3} y + \dots + \alpha_{2m} y^n \quad \dots(5.6)$$

or $$\{\delta\} = \begin{Bmatrix} u(x, y) \\ v(x, y) \end{Bmatrix} = [G]\{\alpha\} = \begin{bmatrix} G_1 & 0 \\ 0 & G_1 \end{bmatrix}\{\alpha\} \quad \dots(5.7)$$

where $G_1 = [1\ x\ y\ x^2\ xy\ y^2\ x^3 \dots y^n]$

$$\{\alpha\}^T = [\alpha_1\ \alpha_2\ \alpha_3\ \alpha_4 \dots \alpha_{2m}]$$

It may be observed that in two dimensional problem, total number of terms m in a complete nth degree polynomial is

$$m = \frac{(n+1)(n+2)}{2} \quad \dots(5.8)$$

For first order complete polynomial $n = 1$,

$$\therefore m = \frac{(1+1)(1+2)}{2} = 3$$

Another convenient way to remember complete two dimensional polynomial is in the form of Pascal Triangle shown in Fig. 5.2

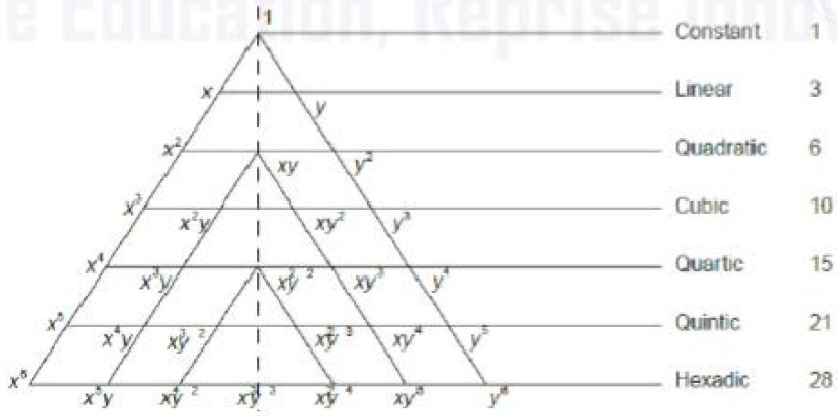


Fig. 5.2 *Pascal triangle*

Three Dimensional Polynomial Shape Function

A general three dimensional shape function of nth order complete polynomial is given by

$$\begin{aligned} u(x, y, z) &= \alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z + \alpha_5 x^2 + \ldots + \alpha_m x^{n-1} z \\ v(x, y, z) &= \alpha_{m+1} + \alpha_{m+2} x + \alpha_{m+3} y + \alpha_{m+4} z + \alpha_{m+5} x^2 + \ldots + \alpha_{2m} x^{n-1} z \\ w(x, y, z) &= \alpha_{2m+1} + \alpha_{2m+2} x + \alpha_{2m+3} y + \alpha_{2m+4} z + \ldots + \alpha_{3m} x^{n-1} z \end{aligned} \qquad \ldots(5.9)$$

or

$$\delta(x, y, z) = \begin{Bmatrix} u(x, y, z) \\ v(x, y, z) \\ w(x, y, z) \end{Bmatrix} = [G]\{\alpha\} = \begin{bmatrix} G_1 & 0 & 0 \\ 0 & G_1 & 0 \\ 0 & 0 & G_1 \end{bmatrix} \{\alpha\} \qquad \ldots(5.10)$$

Where $G_1 = [1\ x\ y\ z\ x^2\ xy\ y^2\ yz\ z^2\ zx\ \ldots z^n\ z^{n-1}x\ \ldots\ zx^{n-1}]$

and

$$\{\alpha\}^T = [\alpha_1\ \alpha_2\ \alpha_3\ \ldots \alpha_{3m}]$$

It may be observed that a complete nth order polynomial in three dimensional case is having number of terms m given by the expression

$$m = \frac{(n+1)(n+2)(n+3)}{6}$$

Thus when $n = 1$, $m = \dfrac{(1+1)(1+2)(1+3)}{6} = 4$

i.e.

$$\alpha_1 + \alpha_2 x + \alpha_3 y + \alpha_4 z$$

5.3 CONVERGENCE REQUIREMENTS OF SHAPE FUNCTIONS

Numerical solutions are approximate solutions. Stiffness coefficients for a displacements model have higher magnitudes compared to those for the exact solutions. In other words the displacements obtained by finite element analysis are lesser than the exact values. Thus the FEM gives lower bound values. Hence it is desirable that as the finite element analysis mesh is refined, the solution approaches the exact values. This requirement is shown graphically in Fig. 5.4. In order to ensure this convergence criteria, the shape functions should satisfy the following requirement:

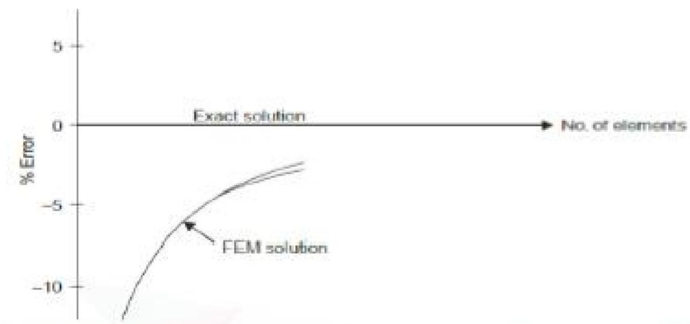


Fig. 5.4 *Convergence of FEM solution*

1. The displacement models must be continuous within the elements and the displacements must be compatible between the adjacent elements. The second part implies that the adjacent elements must deform without causing openings, overlaps or discontinuities between the elements. This requirement is called **'compatibility requirement'**.

   According to Felippa and Clough this requirement is satisfied, if the displacement and its partial derivatives upto one order less than the highest order derivative appearing in strain energy function is continuous. Hence in plane stress and plane strain problems, it is enough if continuity of displacement is satisfied, since strain energy function includes only first order derivatives of the displacement (SE = ½ stress × strain). It implies, it is enough if C^0 continuity is ensured in plane stress and plane strain problems. In case of flexure problems (beams, plates, shells) the strain energy terms include second derivatives of displacements $\left(\text{like } \frac{1}{2}\frac{M^2}{EI} \text{ where } M = -EI\frac{d^2 w}{dx^2}\right)$.

   Hence to satisfy compatibility requirement, not only displacement continuity but slope continuity (C^1 -continuity) should be satisfied. Hence in flexure problems displacements and their first derivatives are selected as nodal field variables.

2. The displacement model should include the **rigid body displacements** of the element. It means in displacement model there should be a term which permit all points on the element to experience the same displacement. It is obvious, if such term do not exists, shifting of the origin of the coordinate system will cause additional stresses and strains, which should not occur. In the displacement model,

$$u = \alpha_1 + \alpha_2 x + \alpha_3 y$$

   the term α_1 provides for the rigid body displacement. Hence to satisfy the requirement of rigid body displacement, there should be constant term in the shape function selected.

3. The displacement models must include the **constant strain state** of the element. This means, there should exist combination of values of polynomial terms that cause all points in the element to experience the same strain. One such combination should occur for each possible strain. The necessity of this requirement is understood physically, if we imagine the refinement of the mesh. As these elements approach infinitesimal size, the strains within the element approach constant values. Unless the shape function term includes these constant strain terms, we cannot hope to converge to a correct solution. In the displacement model,

$$v = \alpha_{m+1} + \alpha_{m+2}\,x + \alpha_{m+3}\,y + \alpha_{m+4}\,x^2 + \ldots + \alpha_{2m}\,y^n$$

α_2 and α_{m+2} provide for uniform strain ε_x,

α_3 and α_{m+3} provide for uniform strain ε_y

An additional consideration in the selection of polynomial shape function for the displacement model is that the pattern should be independent of the orientation of the local coordinate system. This property is known as **Geometric Isotropy, Spatial Isotropy** or **Geometric Invariance.** There are two simple guidelines to construct polynomial series with the desired property of isotropy:

1. Polynomial of order n that are complete, have geometric isotropy.
2. Polynomial of order n that are not complete, yet contain appropriate terms to preserve 'symmetry' have geometric isotropy. The simple test for this property is to interchange x and y in two dimensional problems or x, y, z in cyclic order in three dimensional problems and see that the total expression do not change. However the arbitrary constants may change.

   For example, we wish to construct a cubic polynomial expression for an element that has eight nodal values assigned to it. In this situation, we have to drop two terms from the complete cubic polynomial which contains 10 terms. To maintain geometric isotropy drop only terms that occur in symmetric pairs i.e. x^3, y^3 or x^2y, xy^2. Thus the acceptable eight term cubic polynomials shape function exhibiting geometric isotropy are

   $$\alpha_1 + \alpha_2\,x + \alpha_3\,y + \alpha_4\,x^2 + \alpha_5\,xy + \alpha_6\,y^2 + \alpha_7\,x^2y + \alpha_8\,xy^2$$

   and $\alpha_1 + \alpha_2\,x + \alpha_3\,y + \alpha_4\,x^2 + \alpha_5\,xy + \alpha_6\,y^2 + \alpha_7\,x^3 + \alpha_8\,y^3$

   $$\alpha_1 + \alpha_2\,x + \alpha_3\,y + \alpha_4\,x^2 + \alpha_5\,xy + \alpha_6\,y^2 + \alpha_7\,x^2y + \alpha_8\,xy^2$$

   and $\alpha_1 + \alpha_2\,x + \alpha_3\,y + \alpha_4\,x^2 + \alpha_5\,xy + \alpha_6\,y^2 + \alpha_7\,x^3 + \alpha_8\,y^3$

   In finite element analysis, the safest approach to reach correct solution is to pick the shape functions that satisfy all the requirements. For some problems, however, choosing shape functions that meet all the requirements may be difficult and may involve excessive numerical computations. For this reason some investigators have ventured to formulate shape functions for the elements that do not meet compatibility requirements. In some cases acceptable convergence has been obtained. Such elements are called '**non-conforming elements**'. The main disadvantage of using non-conforming elements is that we no longer know in advance that correct solution is reached.

Characteristic of Shape function

1. Value of shape function of particular node is one and is zero to all other nodes.
2. Sum of all shape function is one.
3. Sum of the derivative of all the shape functions for a particular primary variable is zero.

Coordinate Systems

The following terms are commonly referred in FEM
(i) Global coordinates
(ii) Local coordinates and
(iii) Natural coordinates.
However there is another term '**generalized coordinates**' used for defining a polynomial form of interpolation function. This has nothing to do with the 'coordinates' term used here to define the location of points in the element.

Global coordinates

The coordinate system used to define the points in the entire structure is called global coordinate system. Figure 4.14 shows the cartesian global coordinate system used for some of the typical cases.

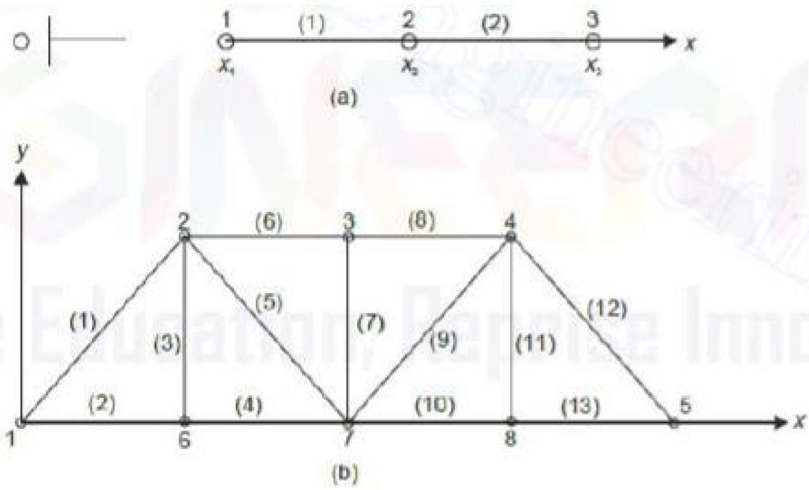


Fig. 4.14 *Global coordinate system*

Local coordinates

For the convenience of deriving element properties, in FEM many times for each element a separate coordinate system is used. For example, for typical elements shown in Fig. 4.14, the local coordinates may be as shown in Fig. 4.15. However the final equations are to be formed in the common coordinate system i.e. global coordinate system only.

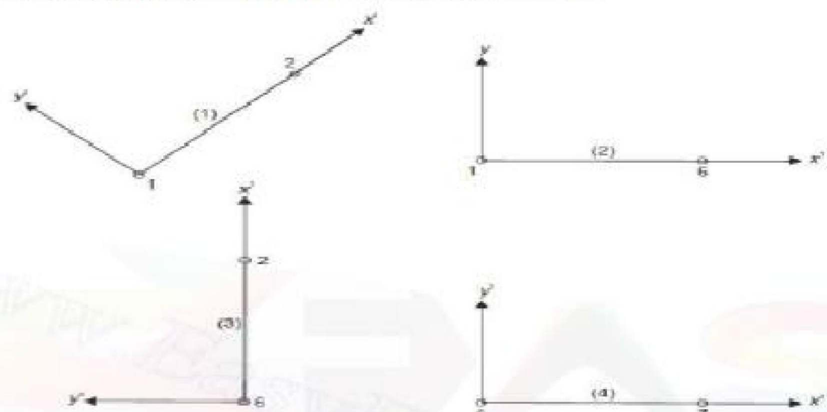


Fig. 4.15 *Local coordinate system*

Natural coordinate

A natural coordinate system is a coordinate system which permits the specification of a point within the element by a set of dimensionless numbers, whose magnitude never exceeds unity. It is obtained by assigning weightages to the nodal coordinates in defining the coordinate of any point inside the element. Hence such system has the property that ith coordinate has unit value at node i of the element and zero value at all other nodes. The use of natural coordinate system is advantages in assembling element properties (stiffness matrices), since closed form integrations formulae are available when the expressions are in natural coordinate systems.

Natural coordinate systems for one dimensional, two dimensional and three dimensional elements are discussed below:

Natural Coordinates in Two Dimensions

Natural coordinates for triangular and rectangular elements are discussed below:

1. Natural Coordinates for Triangular Elements: Consider the typical 3 noded triangular element shown in Fig. 4.20. Since there are three nodes, for any point there are three coordinates, say L_1, L_2 and L_3. From the definition of natural coordinates, we have

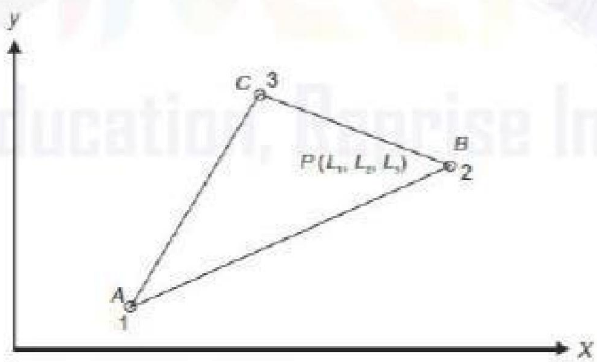


Fig. 4.20 *Typical 3 noded triangular element*

$$L_1 + L_2 + L_3 = 1 \quad \text{...(4.7a)}$$

$$L_1 x_1 + L_2 x_2 + L_3 x_3 = x \quad \text{...(4.7b)}$$

$$L_1 y_1 + L_2 y_2 + L_3 y_3 = y \quad \text{...(4.7c)}$$

Expressing the above equations in matrix form,

$$\begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{bmatrix} \begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} = \begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix}$$

$$\begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} = \begin{bmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{bmatrix}^{-1} \begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix}$$

It can be shown that the determinant,

$$\begin{vmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{vmatrix}$$

Proof: Now,

$$\text{Det} = \begin{vmatrix} 1 & 1 & 1 \\ x_1 & x_2 & x_3 \\ y_1 & y_2 & y_3 \end{vmatrix} = (x_2 y_3 - x_3 y_2) - (x_1 y_3 - x_3 y_1) + (x_1 y_2 - x_2 y_1)$$

Consider the triangle *ABC* shown in Fig. 4.21. Drop perpendiculars *AD*, *BE* and *CF* on to x-axis.

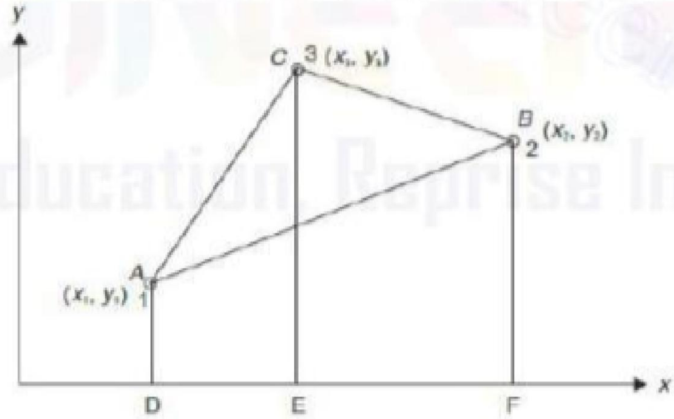


Fig. 4.21

Now, Area of triangle ABC

=Area *ADEC* + Area *CEFB* – Area *ADFB*

$$= \frac{1}{2}(AD + CE)\,DE + \frac{1}{2}(CE + BF)EF - \frac{1}{2}(AD + BF)\,DF$$

$$= \frac{1}{2}(y_1 + y_3)(x_3 - x_1) + \frac{1}{2}(y_3 + y_2)(x_2 - x_3) - \frac{1}{2}(y_1 + y_2)(x_2 - x_1)$$

$$= \frac{1}{2}\left[y_1x_3 - y_1x_1 + y_3x_3 - y_3x_1 + y_3x_2 - y_3x_3 + y_2x_2 - y_2x_3 - y_1x_2 + y_1x_1 - y_2x_2 + y_2x_1\right]$$

$$= \frac{1}{2}\left[y_1x_3 - y_3x_1 + y_3x_2 - y_2x_3 - y_1x_2 + y_2x_1\right]$$

$$= \frac{1}{2}\left[(x_2y_3 - x_3y_2) - (x_1y_3 - x_3y_1) + (x_1y_2 - x_2y_1)\right]$$

$$= \frac{1}{2}\,Det$$

$\therefore$ Det = 2 Area of triangle *ABC* = 2*A* ...(4.8)

$$\therefore \begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} = \frac{1}{2A}\begin{bmatrix} x_2y_3 - x_3y_2 & x_3y_1 - x_1y_3 & x_1y_2 - x_2y_1 \\ y_2 - y_3 & y_3 - y_1 & y_1 - y_2 \\ x_3 - x_2 & x_1 - x_3 & x_2 - x_1 \end{bmatrix}^T \begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix}$$

$$= \frac{1}{2A}\begin{bmatrix} x_2y_3 - x_3y_2 & y_2 - y_3 & x_3 - x_2 \\ x_3y_1 - x_1y_3 & y_3 - y_1 & x_1 - x_3 \\ x_1y_2 - x_2y_1 & y_1 - y_2 & x_2 - x_1 \end{bmatrix}\begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix} = \frac{1}{2A}\begin{bmatrix} a_1 & b_1 & c_1 \\ a_2 & b_2 & c_2 \\ a_3 & b_3 & c_3 \end{bmatrix}\begin{Bmatrix} 1 \\ x \\ y \end{Bmatrix}$$

where $a_1 = x_2y_3 - x_3y_2 \quad a_2 = x_3y_1 - x_1y_3 \quad a_3 = x_1y_2 - x_2y_1$

$b_1 = y_2 - y_3 \quad b_2 = y_3 - y_1 \quad b_3 = y_1 - y_2$

$c_1 = x_3 - x_2 \quad c_2 = x_1 - x_3 \quad c_3 = x_2 - x_1$

[Note the cyclic order of subscript and absence of subscript of left hand term in right hand terms]

Thus

$$\begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} = \begin{Bmatrix} \dfrac{a_1 + b_1x + c_1y}{2A} \\ \dfrac{a_2 + b_2x + c_2y}{2A} \\ \dfrac{a_3 + b_3x + c_3y}{2A} \end{Bmatrix} \qquad \ldots(4.9)$$

Referring to Fig. 4.22 and applying equation 4.8, we get Area of subtraingle *CPB*

$$= 2A_1 = \begin{bmatrix} 1 & 1 & 1 \\ x & x_2 & x_3 \\ y & y_2 & y_3 \end{bmatrix}$$

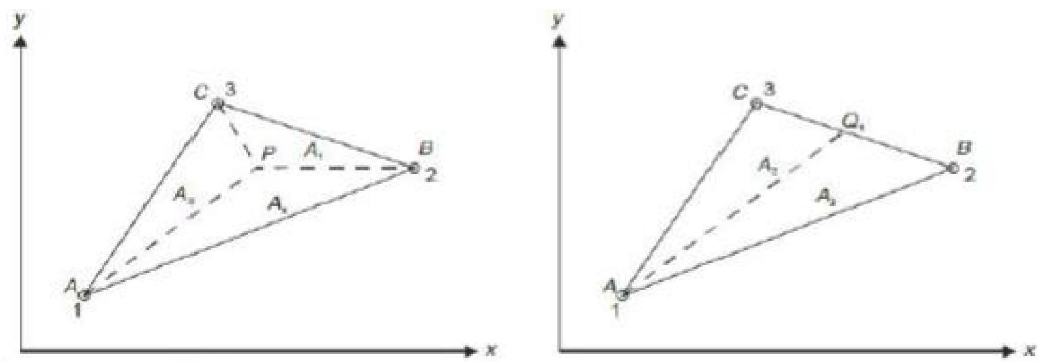


Fig. 4.22 *Area coordinates for a triangle*

i.e., $2A_1 = x_2y_3 - x_3y_2 - (xy_3 - x_3y) + xy_2 - x_2y$

$= x_2y_3 - x_3y_2 + x(y_2 - y_3) + y(x_3 - x_2)$

$= a_1 + b_1x + c_1y$

Thirdly $2A_2 = a_2 + b_2x + c_2y$

and $2A_3 = a_3 + b_3x + c_3y$

$\therefore$ Equation 4.9 reduces to

$$\begin{Bmatrix} L_1 \\ L_2 \\ L_3 \end{Bmatrix} = \begin{Bmatrix} \dfrac{2A_1}{2A} \\ \dfrac{2A_2}{2A} \\ \dfrac{2A_3}{2A} \end{Bmatrix} = \frac{1}{A}\begin{Bmatrix} A_1 \\ A_2 \\ A_3 \end{Bmatrix} \qquad \ldots(4.10)$$

where A_1, A_2 and A_3 are the areas of sub-triangles *PCB*, *PAC* and *PAB*, which are opposite to nodes 1, 2 and 3 respectively. Hence the natural coordinates in triangles are also known as **area coordinates**.

Natural Coordinates in Three Dimensions

Natural coordinates for a 4 noded tetrahedron may be derived and it results into volume coordinates. Consider the typical tetrahedron shown in Fig. 4.24.

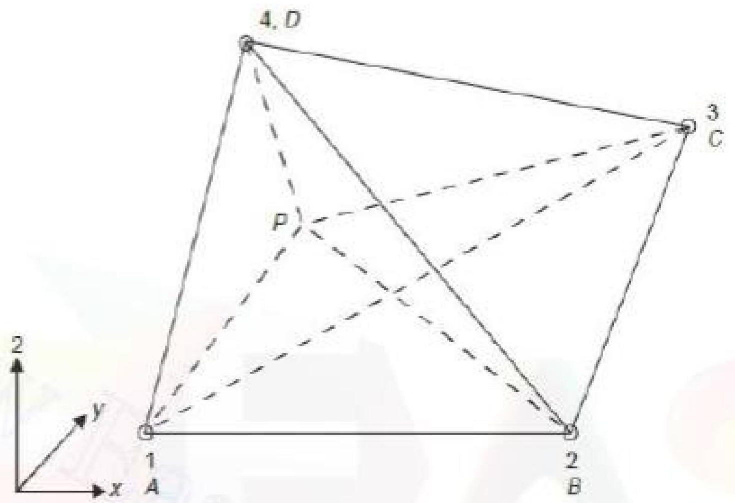


Fig. 4.24 *Tetrahedron coordinates*

The natural coordinates are related to the Cartesian coordinates as follows:

$$\begin{Bmatrix} 1 \\ x \\ y \\ z \end{Bmatrix} = \begin{bmatrix} 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 \\ y_1 & y_2 & y_3 & y_4 \\ z_1 & z_2 & z_3 & z_4 \end{bmatrix} \begin{Bmatrix} L_1 \\ L_2 \\ L_3 \\ L_4 \end{Bmatrix} \qquad \ldots(4.12)$$

The above equation may be solved by inverting the 4 × 4 matrix. It gives

$$L_i = \frac{1}{6V}\,(a_i + b_i x + c_i y + d_i z), \text{ for } i = 1, 2, 3 \text{ and } 4 \qquad \ldots(4.13)$$

where $6V = \begin{vmatrix} 1 & 1 & 1 & 1 \\ x_1 & x_2 & x_3 & x_4 \\ y_1 & y_2 & y_3 & y_4 \\ z_1 & z_2 & z_3 & z_4 \end{vmatrix} = 6 \times$ volume of tetrahedron defined by nodes 1, 2, 3 and 4

and

$$a_1 = \begin{vmatrix} x_2 & x_3 & x_4 \\ y_2 & y_3 & y_4 \\ z_2 & z_3 & z_4 \end{vmatrix} \quad b_1 = \begin{vmatrix} 1 & 1 & 1 \\ y_2 & y_3 & y_4 \\ z_2 & z_3 & z_4 \end{vmatrix}$$

$$c_1 = \begin{vmatrix} 1 & 1 & 1 \\ x_2 & x_3 & x_4 \\ z_2 & z_3 & z_4 \end{vmatrix} \text{ and } d_1 = \begin{vmatrix} 1 & 1 & 1 \\ x_2 & x_3 & x_4 \\ y_2 & y_3 & y_4 \end{vmatrix}$$

The other constants are obtained by cyclic changes in the subscripts. It may be noted that the above equations are valid only when the nodes are numbered so that nodes 1, 2 and 3 are ordered counter clockwise when viewed from node 4. It is also necessary that for coordinates system of right hand rule is strictly adhered to.

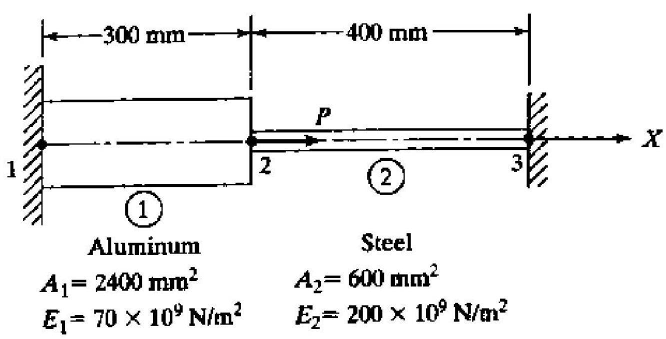
If V_i is the volume of the smaller tetrahedron which has vertices P and the three nodes other than the node i, then the tetrahedron coordinates can be considered as volume coordinates, defined as

$$L_i = \frac{V_i}{V} \quad \text{for } i = 1, 2, 3 \text{ and } 4 \qquad \ldots(4.14)$$

Example 3.4

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.



Solution

(a) The element stiffness matrices are

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

and

$$\begin{array}{cc} & \begin{array}{cc} 2 & 3 \end{array} \\ \mathbf{k}^2 = \dfrac{200 \times 10^3 \times 600}{400} & \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \end{array}$$

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

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

The global load vector is

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

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. 3.80, 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

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

(b) The element stresses (Eq. 3.16) 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 3.5

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.

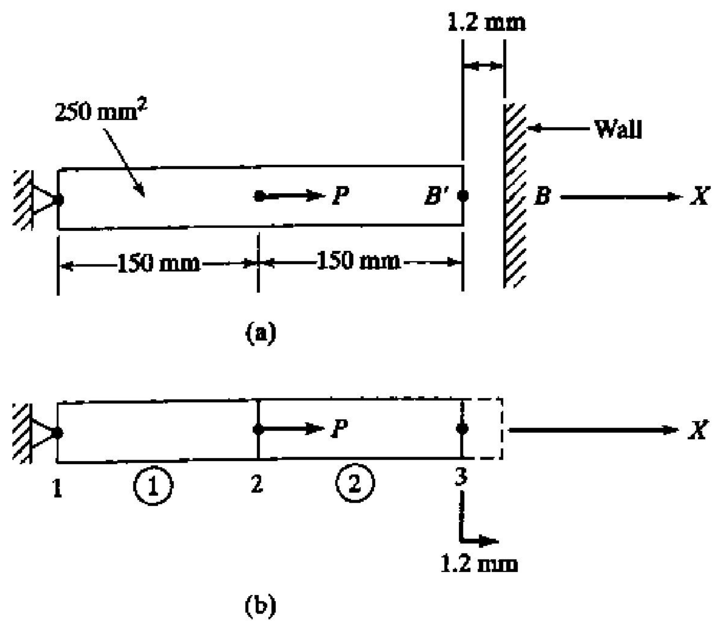


FIGURE E3.5

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 re-solved, 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. 3.5b. 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

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

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 $C = (2/3) \times 10^{10}$, 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

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

The element stresses are

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

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

The reaction forces are

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

and

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

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. ■

Truss

INTRODUCTION

The finite element analysis of truss structures is presented in this chapter. Two-dimensional trusses (or plane trusses) are treated in Section 4.2. In Section 4.3, this treatment is readily generalized to handle three-dimensional trusses. A typical plane truss is shown in Fig. 4.1. A truss structure consists only of two-force members. That is, every truss element is in direct tension or compression (Fig. 4.2). In a truss, it is required that all loads and reactions are applied only at the joints and that all members are connected together at their ends by frictionless pin joints. Every engineering student has, in a course on statics, analyzed trusses using the method of joints and the method of sections. These methods, while illustrating the fundamentals of statics, become tedious when applied to large-scale statically indeterminate truss structures. Further, joint displacements are not readily obtainable. The finite element method on the other hand is applicable to statically determinate or indeterminate structures alike. The finite element method also provides joint deflections. Effects of temperature changes and support settlements can also be routinely handled.

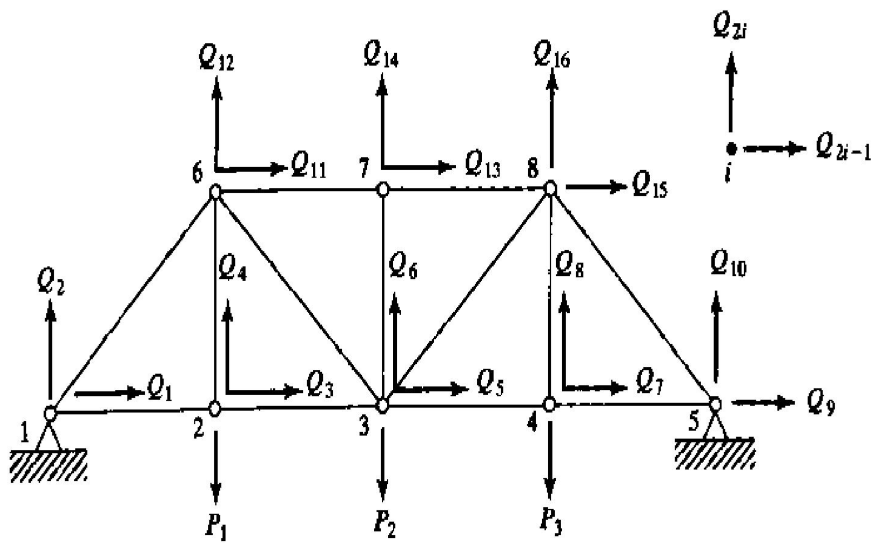


FIGURE 4.1 A two-dimensional truss.

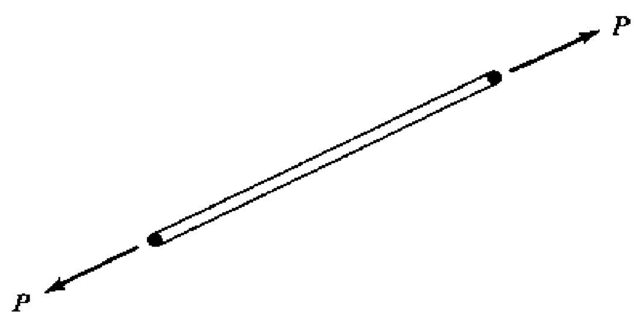


FIGURE 4.2 A two-force member.

PLANE TRUSSES

Modeling aspects discussed in Chapter 3 are now extended to the two-dimensional truss. The steps involved are discussed here.

Local and Global Coordinate Systems

The main difference between the one-dimensional structures considered in Chapter 3 and trusses is that the elements of a truss have various orientations. To account for these different orientations, **local** and **global** coordinate systems are introduced as follows:

A typical plane-truss element is shown in local and global coordinate systems in Fig. 4.3. In the local numbering scheme, the two nodes of the element are numbered 1 and 2. The local coordinate system consists of the x'-axis, which runs along the element from node 1 toward node 2. All quantities in the local coordinate system will be denoted by a prime ('). The global x-, y-coordinate system is fixed and does not depend on the orientation of the element. Note that x, y, and z form a right-handed coordinate system with the z-axis coming straight out of the paper. In the global coordinate system,

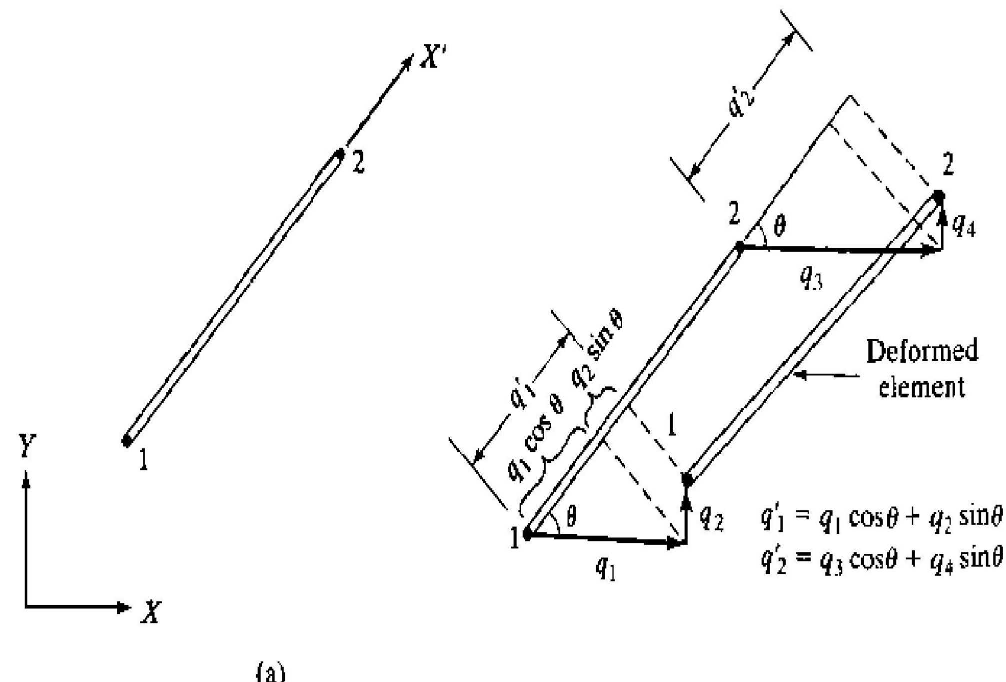


FIGURE 4.3 A two-dimensional truss element in (a) a local coordinate system and (b) a global coordinate system.

every node has two degrees of freedom (dofs). A systematic numbering scheme is adopted here: A node whose global node number is j has associated with it dofs $2j - 1$ and $2j$. Further, the global displacements associated with node j are Q_{2j-1} and Q_{2j}, as shown in Fig. 4.1.

Let q'_1 and q'_2 be the displacements of nodes 1 and 2, respectively, in the local coordinate system. Thus, the element displacement vector in the local coordinate system is denoted by

$$\mathbf{q}' = [q'_1, q'_2]^T \tag{4.1}$$

The element displacement vector in the global coordinate system is a (4×1) vector denoted by

$$\mathbf{q} = [q_1, q_2, q_3, q_4]^T \tag{4.2}$$

The relationship between $\mathbf{q}'$ and $\mathbf{q}$ is developed as follows: In Fig. 4.3b, we see that q'_1 equals the sum of the projections of q_1 and q_2 onto the x'-axis. Thus,

$$q'_1 = q_1 \cos\theta + q_2 \sin\theta \tag{4.3a}$$

Similarly,

$$q'_2 = q_3 \cos\theta + q_4 \sin\theta \tag{4.3b}$$

At this stage, the **direction cosines** ℓ and m are introduced as $\ell = \cos\theta$ and $m = \cos\phi$ $(= \sin\theta)$. These direction cosines are the cosines of the angles that the local x'-axis makes with the global x-, y-axes, respectively. Equations 4.3a and 4.3b can now be written in matrix form as

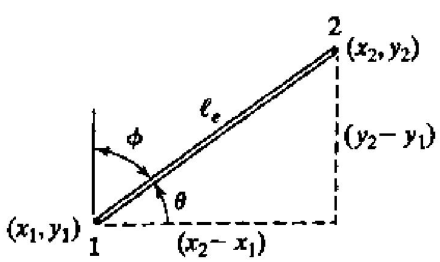
$$\mathbf{q}' = \mathbf{Lq} \tag{4.4}$$

where the **transformation matrix** $\mathbf{L}$ is given by

$$\mathbf{L} = \begin{bmatrix} \ell & m & 0 & 0 \\ 0 & 0 & \ell & m \end{bmatrix} \tag{4.5}$$

Formulas for Calculating ℓ and m

Simple formulas are now given for calculating the direction cosines ℓ and m from nodal coordinate data. Referring to Fig. 4.4, let (x_1, y_1) and (x_2, y_2) be the coordinates of nodes 1 and 2, respectively. We then have



$$\ell = \cos\theta = \frac{x_2 - x_1}{\ell_e}$$

$$m = \cos\phi = \frac{y_2 - y_1}{\ell_e} (= \sin\theta)$$

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

FIGURE 4.4 Direction cosines

$$\ell = \frac{x_2 - x_1}{\ell_e} \qquad m = \frac{y_2 - y_1}{\ell_e} \tag{4.6}$$

where the length ℓ_e is obtained from

$$\ell_e = \sqrt{(x_2 - x_1)^2 + (y_2 - y_1)^2} \tag{4.7}$$

The expressions in Eqs. 4.6 and 4.7 are obtained from nodal coordinate data and can readily be implemented in a computer program.

Element Stiffness Matrix

An important observation will now be made: *The truss element is a one-dimensional element when viewed in the local coordinate system.* This observation allows us to use previously developed results in Chapter 3 for one-dimensional elements. Consequently, from Eq. 3.26, the element stiffness matrix for a truss element in the local coordinate system is given by

$$\mathbf{k} = \frac{E_e A_e}{\ell_e}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{4.8}$$

where A_e is the element cross-sectional area and E_e is Young's modulus. The problem at hand is to develop an expression for the element stiffness matrix in the global coordinate system. This is obtainable by considering the strain energy in the element. Specifically, the element strain energy in local coordinates is given by

$$U_e = \tfrac{1}{2}\mathbf{q}'^{\mathrm{T}}\mathbf{k}'\mathbf{q}' \tag{4.9}$$

Substituting for $\mathbf{q}' = \mathbf{Lq}$ into Eq. 4.9, we get

$$U_e = \tfrac{1}{2}\mathbf{q}^{\mathrm{T}}[\mathbf{L}^{\mathrm{T}}\mathbf{k}'\mathbf{L}]\mathbf{q} \tag{4.10}$$

The strain energy in global coordinates can be written as

$$U_e = \tfrac{1}{2}\mathbf{q}^{\mathrm{T}}\mathbf{k}\mathbf{q} \tag{4.11}$$

where $\mathbf{k}$ is the element stiffness matrix in global coordinates. From the previous equation, we obtain the element stiffness matrix in global coordinates as

$$\mathbf{k} = \mathbf{L}^{\mathrm{T}}\mathbf{k}'\mathbf{L} \tag{4.12}$$

Substituting for $\mathbf{L}$ from Eq. 4.5 and for $\mathbf{k}'$ from Eq. 4.8, we get

$$\mathbf{k} = \frac{E_e A_e}{\ell_e}\begin{bmatrix} \ell^2 & \ell m & -\ell^2 & -\ell m \\ \ell m & m^2 & -\ell m & -m^2 \\ -\ell^2 & -\ell m & \ell^2 & \ell m \\ -\ell m & -m^2 & \ell m & m^2 \end{bmatrix} \tag{4.13}$$

The element stiffness matrices are assembled in the usual manner to obtain the structural stiffness matrix. This assembly is illustrated in Example 4.1. The computer logic for directly placing element stiffness matrices into global matrices for banded and skyline solutions is explained in Section 4.4.

The derivation of the result $\mathbf{k} = \mathbf{L}^{\mathrm{T}}\mathbf{k}'\mathbf{L}$ also follows from Galerkin's variational principle. The virtual work δW as a result of virtual displacement $\boldsymbol{\psi}'$ is

$$\delta W = \boldsymbol{\psi}'^{\mathrm{T}}(\mathbf{k}'\mathbf{q}') \tag{4.14a}$$

Since $\boldsymbol{\psi}' = \mathbf{L}\boldsymbol{\psi}$ and $\mathbf{q}' = \mathbf{L}\mathbf{q}$, we have

$$\begin{aligned}\delta W &= \boldsymbol{\psi}^{\mathrm{T}}[\mathbf{L}^{\mathrm{T}}\mathbf{k}'\mathbf{L}]\mathbf{q} \\ &= \boldsymbol{\psi}^{\mathrm{T}}\mathbf{k}\mathbf{q}\end{aligned} \tag{4.14b}$$

Stress Calculations

Expressions for the element stresses can be obtained by noting that a truss element in local coordinates is a simple two-force member (Fig. 4.2). Thus, the stress σ in a truss element is given by

$$\sigma = E_e \epsilon \tag{4.15a}$$

Since the strain ϵ is the change in length per unit original length,

$$\begin{aligned}\sigma &= E_e \frac{q_2' - q_1'}{\ell_e} \\ &= \frac{E_e}{\ell_e}[-1 \quad 1]\begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix}\end{aligned} \tag{4.15b}$$

This equation can be written in terms of the global displacements $\mathbf{q}$ using the transformation $\mathbf{q}' = \mathbf{L}\mathbf{q}$ as

$$\sigma = \frac{E_e}{\ell_e}[-1 \quad 1]\mathbf{L}\mathbf{q} \tag{4.15c}$$

Substituting for $\mathbf{L}$ from Eq. 4.5 yields

$$\sigma = \frac{E_e}{\ell_e}[-\ell \quad -m \quad \ell \quad m]\mathbf{q} \tag{4.16}$$

Once the displacements are determined by solving the finite element equations, the stresses can be recovered from Eq. 4.16 for each element. Note that a positive stress implies that the element is in tension and a negative stress implies compression.

Example 4.1

Consider the four-bar truss shown in Fig. E4.1a. 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.

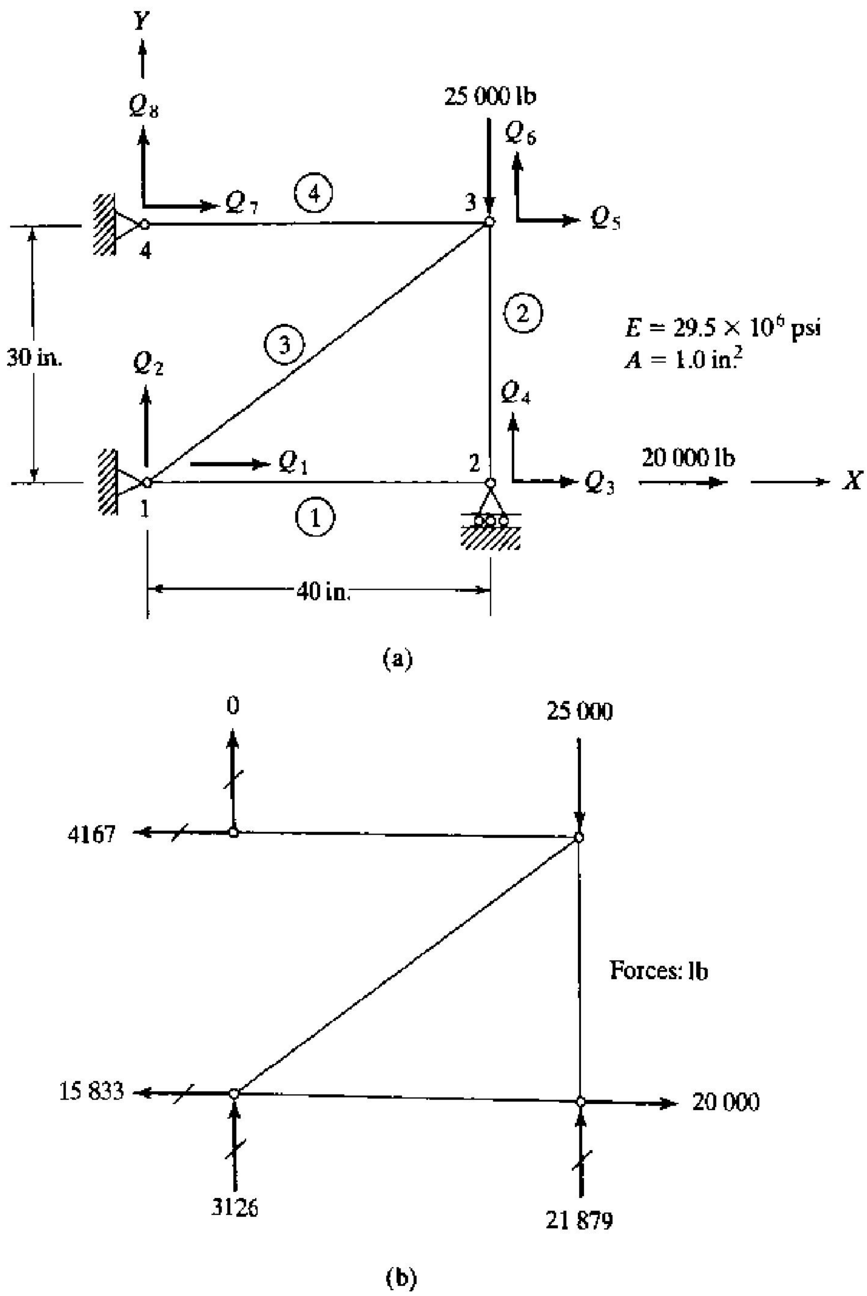


FIGURE E4.1

Solution

(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 in Eqs. 4.6 and 4.7, 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 $\ell = (x_3 - x_1)/\ell_e = (40 - 0)/50 = 0.8$ and $m = (y_3 - y_1)/\ell_e = (30 - 0)/50 = 0.6$. Now, using Eq. 4.13, the element stiffness matrices for element 1 can be written as

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

The global dofs associated with element 1, which is connected between nodes 1 and 2, are indicated in $\mathbf{k}^1$ earlier. These global dofs are shown in Fig. E4.1a and assist in assembling the various element stiffness matrices.

The element stiffness matrices of elements 2, 3, and 4 are as follows:

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

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

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

(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

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

(c) The structural stiffness matrix **K** given above needs to be modified to account for the boundary conditions. The elimination approach discussed in Chapter 3 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

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

(d) The stress in each element can now be determined from Eq. 4.16, as shown below.

The connectivity of element 1 is 1 − 2. Consequently, the nodal displacement vector for element 1 is given by $\mathbf{q} = [0, 0, 27.12 \times 10^{-3}, 0]^{\mathrm{T}}$, and Eq. 4.16 yields

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

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

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

$$\sigma_3 = -5208.0 \text{ psi}$$
$$\sigma_4 = 4167.0 \text{ psi}$$

(e) The final step is to determine the support reactions. We need to determine the reaction forces along dofs 1, 2, 4, 7, and 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 these 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

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

A free body diagram of the truss with reaction forces and applied loads is shown in Fig. E4.1b. ■