CHAPTER 3

Truss Structures: The Direct Stiffness Method

3.1 INTRODUCTION

The simple line elements discussed in Chapter 2 introduced the concepts of nodes, nodal displacements, and element stiffness matrices. In this chapter, creation of a finite element model of a mechanical system composed of any number of elements is considered. The discussion is limited to *truss structures,* which we define as structures composed of straight elastic members subjected to axial forces only. Satisfaction of this restriction requires that all members of the truss be bar elements and that the elements be connected by pin joints such that each element is free to rotate about the joint. Although the bar element is inherently one dimensional, it is quite effectively used in analyzing both two- and three-dimensional trusses, as is shown.

The *global* coordinate system is the reference frame in which displacements of the structure are expressed and usually chosen by convenience in consideration of overall geometry. Considering the simple cantilever truss shown in Figure 3.1a, it is logical to select the global *XY* axes as parallel to the predominant geometric "axes" of the truss as shown. If we examine the circled joint, for example, redrawn in Figure 3.1b, we observe that five *element nodes* are physically connected at one *global node* and the *element x* axes do not coincide with the *global X* axis. The physical connection and varying geometric orientation of the elements lead to the following premises inherent to the finite element method:

1. The element nodal displacement of each connected element must be the same as the displacement of the connection node in the global coordinate system; the mathematical formulation, as will be seen, enforces this requirement (displacement compatibility).

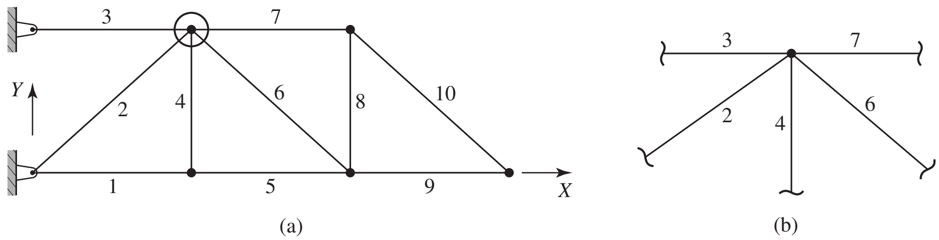


Figure 3.1
(a) Two-dimensional truss composed of ten elements. (b) Truss joint connecting five elements.

2. The physical characteristics (in this case, the stiffness matrix and element force) of each element must be transformed, mathematically, to the global coordinate system to represent the structural properties in the global system in a consistent mathematical frame of reference.
3. The individual element parameters of concern (for the bar element, axial stress) are determined after solution of the problem in the global coordinate system by transformation of results back to the element reference frame (postprocessing).

Why are we basing the formulation on displacements? Generally, a design engineer is more interested in the stress to which each truss member is subjected, to compare the stress value to a known material property, such as the yield strength of the material. Comparison of computed stress values to material properties may lead to changes in material or geometric properties of individual elements (in the case of the bar element, the cross-sectional area). The answer to the question lies in the nature of physical problems. It is much easier to predict the loading (forces and moments) to which a structure is subjected than the deflections of such a structure. If the external loads are specified, the relations between loads and displacements are formulated in terms of the stiffness matrix and we solve for displacements. Back-substitution of displacements into individual element equations then gives us the strains and stresses in each element as desired. This is the *stiffness* method and is used exclusively in this text. In the alternate procedure, known as the *flexibility* method [1], displacements are taken as the known quantities and the problem is formulated such that the forces (more generally, the stress components) are the unknown variables. Similar discussion applies to nonstructural problems. In a heat transfer situation, the engineer is most often interested in the rate of heat flow into, or out of, a particular device. While temperature is certainly of concern, temperature is not the primary variable of interest. Nevertheless, heat transfer problems are generally formulated such that temperature is the primary dependent variable and heat flow is a secondary, computed variable in analogy with strain and stress in structural problems.

Returning to consideration of Figure 3.1b, where multiple elements are connected at a global node, the geometry of the connection determines the relations between element displacements and global displacements as well as the contributions of individual elements to overall structural stiffness. In the *direct stiffness* method, the stiffness matrix of each element is transformed from the element coordinate system to the global coordinate system. The individual terms of each transformed element stiffness matrix are then added directly to the global stiffness matrix as determined by element connectivity (as noted, the connectivity relations ensure compatibility of displacements at joints and nodes where elements are connected). For example and simply by intuition at this point, elements 3 and 7 in Figure 3.1b should contribute stiffness only in the global X direction; elements 2 and 6 should contribute stiffness in both X and Y global directions; element 4 should contribute stiffness only in the global Y direction. The element transformation and stiffness matrix assembly procedures to be developed in this chapter indeed verify the intuitive arguments just made.

The direct stiffness assembly procedure, subsequently described, results in exactly the same system of equations as would be obtained by a formal equilibrium approach. By a *formal equilibrium approach,* we mean that the equilibrium equations for each joint (node) in the structure are explicitly expressed, including deformation effects. This should *not* be confused with the method of joints [2], which results in computation of forces only and does not take displacement into account. Certainly, if the force in each member is known, the physical properties of the member can be used to compute displacement. However, enforcing compatibility of displacements at connections (global nodes) is algebraically tedious. Hence, we have another argument for the stiffness method: Displacement compatibility is assured via the formulation procedure. Granted that we have to "backtrack" to obtain the information of true interest (strain, stress), but the backtracking is algebraic and straightforward, as will be illustrated.

3.2 NODAL EQUILIBRIUM EQUATIONS

To illustrate the required conversion of element properties to a global coordinate system, we consider the one-dimensional bar element as a structural member of a two-dimensional truss. Via this relatively simple example, the *assembly* procedure of essentially any finite element problem formulation is illustrated. We choose the element type (in this case we have only one selection, the bar element); specify the geometry of the problem (element connectivity); formulate the algebraic equations governing the problem (in this case, static equilibrium); specify the boundary conditions (known displacements and applied external forces); solve the system of equations for the global displacements; and back-substitute displacement values to obtain *secondary* variables, including strain, stress, and reaction forces at constrained locations (boundary conditions). The reader is advised to note that we use the term *secondary* variable only in the mathematical sense; strain and stress are secondary only in the sense that the values are computed after the general solution for displacements. The strain and stress values are of *primary importance* in design.

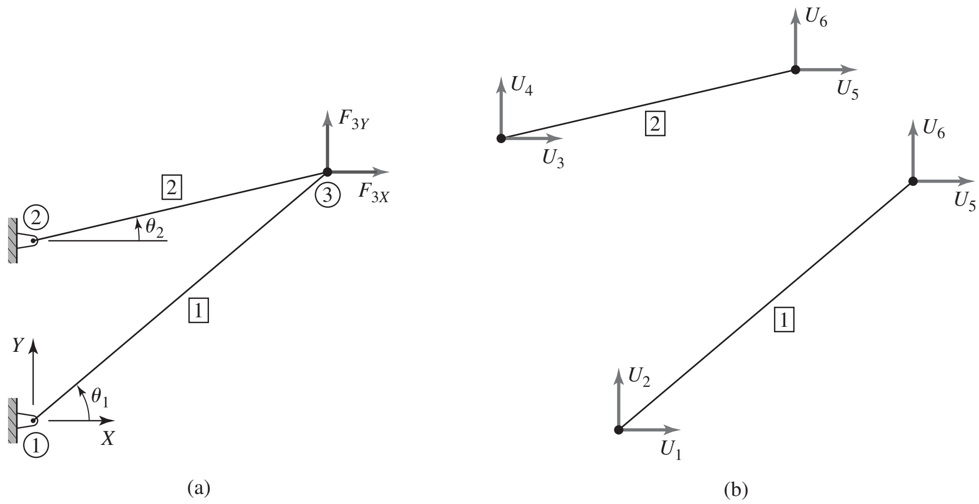


Figure 3.2
(a) A two-element truss with node and element numbers. (b) Global displacement notation.

Conversion of element equations from element coordinates to global coordinates and assembly of the global equilibrium equations are described first in the two-dimensional case with reference to Figure 3.2a. The figure depicts a simple two-dimensional truss composed of two structural members joined by pin connections and subjected to applied external forces. The pin connections are taken as the nodes of two bar elements as shown; node and element numbers, as well as the selected global coordinate system are also shown. The corresponding global displacements are shown in Figure 3.2b. The convention used here for global displacements is that U_{2i-1} is displacement in the global X direction of node i and U_{2i} is displacement of node i in the global Y direction. The convention is by no means restrictive; the convention is selected such that displacements in the direction of the global X axis are odd numbered and displacements in the direction of the global Y axis are even numbered. (In using FEM software, the reader will find that displacements are denoted in various fashions, UX, UY, UZ, etc.) Orientation angle θ for each element is measured as positive from the global X axis to the element x axis, as shown. Node numbers are circled while element numbers are in boxes. Element numbers are superscripted in the notation.

To obtain the equilibrium conditions, free-body diagrams of the three connecting nodes and the two elements are drawn in Figure 3.3. Note that the external forces are numbered via the same convention as the global displacements. For node 1, (Figure 3.3a), we have the following equilibrium equations in the global X and Y directions, respectively:

$$F_1 - f_1^{(1)} \cos\theta_1 = 0 \tag{3.1a}$$

$$F_2 - f_1^{(1)} \sin\theta_1 = 0 \tag{3.1b}$$

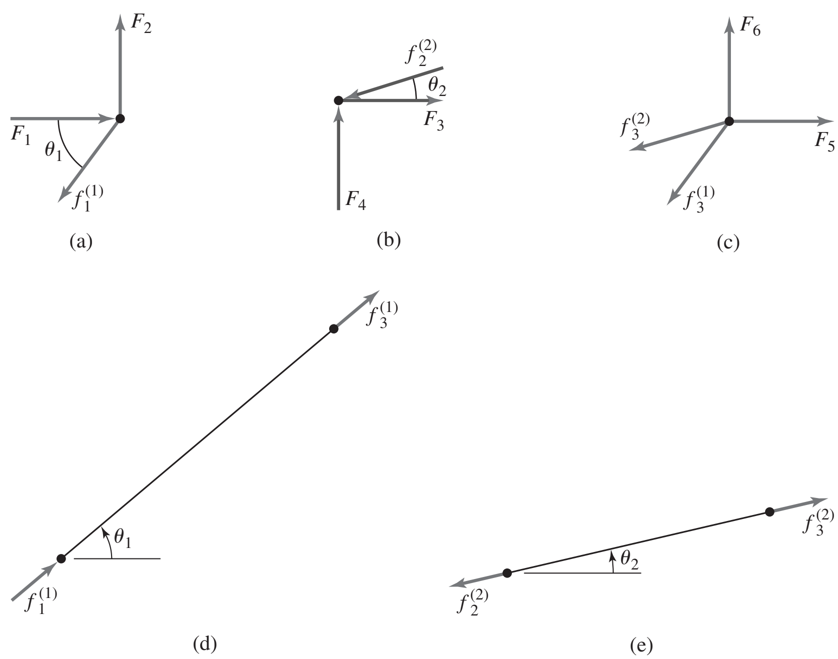


Figure 3.3
(a)–(c) Nodal free-body diagrams. (d) and (e) Element free-body diagrams.

and for node 2,

$$F_3 - f_2^{(2)} \cos \theta_2 = 0 \tag{3.2a}$$

$$F_4 - f_2^{(2)} \sin \theta_2 = 0 \tag{3.2b}$$

while for node 3,

$$F_5 - f_3^{(1)} \cos \theta_1 - f_3^{(2)} \cos \theta_2 = 0 \tag{3.3a}$$

$$F_6 - f_3^{(1)} \sin \theta_1 - f_3^{(2)} \sin \theta_2 = 0 \tag{3.3b}$$

Equations 3.1–3.3 simply represent the conditions of static equilibrium from a rigid body mechanics standpoint. Assuming external loads F_5 and F_6 are known, these six nodal equilibrium equations formally contain eight unknowns (forces). Since the example truss is statically determinate, we can invoke the additional equilibrium conditions applicable to the truss as a whole as well as those for the individual elements (Figures 3.3d and 3.3e) and eventually solve for all of the forces. However, a more systematic procedure is obtained if the formulation is transformed so that the unknowns are nodal displacements. Once the transformation is accomplished, we find that the number of unknowns is exactly the same as the number of nodal equilibrium equations. In addition, static *indeterminacy* is automatically accommodated. As the reader may recall from study of mechanics of materials, the solution of statically indeterminate systems requires

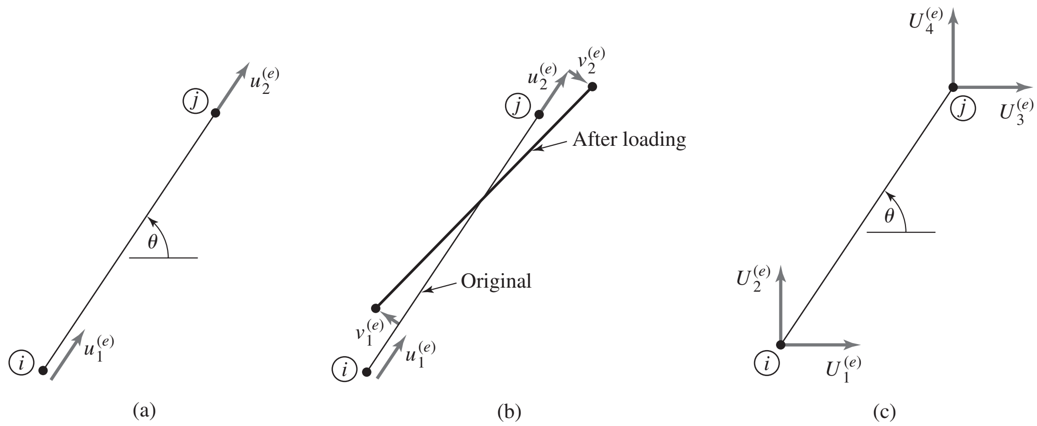


Figure 3.4
(a) Bar element at orientation θ. (b) General displacements of a bar element. (c) Bar element global displacements.

specification of one or more displacement relations; hence, the displacement formulation of the finite element method includes such situations.

To illustrate the transformation to displacements, Figure 3.4a depicts a bar element connected at nodes i and j in a general position in a two-dimensional (2-D) truss structure. As a result of external loading on the truss, we assume that nodes i and j undergo 2-D displacement, as shown in Figure 3.4b. Since the element must remain connected at the structural joints, the connected element nodes must undergo the same 2-D displacements. This means that the element is subjected not only to axial motion but rotation as well. To account for the rotation, we added displacements v_1 and v_2 at element nodes 1 and 2, respectively, in the direction perpendicular to the element x axis. Owing to the assumption of smooth pin joint connections, the perpendicular displacements are not associated with element stiffness; nevertheless, these displacements must exist so that the element remains connected to the structural joint so that the element displacements are compatible with (i.e., the same as) joint displacements. Although the element undergoes a rotation in general, for computation purposes, orientation angle θ is assumed to be the same as in the undeformed structure. This is a result of the assumption of small, elastic deformations and is used throughout the text.

To now relate element nodal displacements referred to the element coordinates to element displacements in global coordinates, Figure 3.4c shows element nodal displacements in the global system using the notation

$$U_1^{(e)} = \text{element node 1 displacement in the global } X \text{ direction}$$

$$U_2^{(e)} = \text{element node 1 displacement in the global } Y \text{ direction}$$

$$U_3^{(e)} = \text{element node 2 displacement in the global } X \text{ direction}$$

$$U_4^{(e)} = \text{element node 2 displacement in the global } Y \text{ direction}$$

Again, note the use of capital letters for global quantities and the superscript notation to refer to an individual element. As the nodal displacements must be the same in both coordinate systems, we can equate vector components of global displacements to element system displacements to obtain the relations

$$\begin{aligned} u_1^{(e)} &= U_1^{(e)}\cos\theta + U_2^{(e)}\sin\theta \\ v_1^{(e)} &= -U_1^{(e)}\sin\theta + U_2^{(e)}\cos\theta \end{aligned} \tag{3.4a}$$

$$\begin{aligned} u_2^{(e)} &= U_3^{(e)}\cos\theta + U_4^{(e)}\sin\theta \\ v_1^{(e)} &= -U_3^{(e)}\sin\theta + U_4^{(e)}\cos\theta \end{aligned} \tag{3.4b}$$

As noted, the v displacement components are not associated with element stiffness, hence not associated with element forces, so we can express the axial deformation of the element as

$$\delta^{(e)} = u_2^{(e)} - u_1^{(e)} = \left(U_3^{(e)} - U_1^{(e)}\right)\cos\theta + \left(U_4^{(e)} - U_2^{(e)}\right)\sin\theta \tag{3.5}$$

The net axial force acting on the element is then

$$f^{(e)} = k^{(e)}\delta^{(e)} = k^{(e)}\left\{\left(U_3^{(e)} - U_1^{(e)}\right)\cos\theta + \left(U_4^{(e)} - U_2^{(e)}\right)\sin\theta\right\} \tag{3.6}$$

Utilizing Equation 3.6 for element 1 (Figure 3.3d) while noting that the displacements of element 1 are related to the specified global displacements as $U_1^{(1)} = U_1$, $U_2^{(1)} = U_2$, $U_3^{(1)} = U_5$, $U_4^{(1)} = U_6$, we have the force in element 1 as

$$f_3^{(1)} = -f_1^{(1)} = k^{(1)}[(U_5 - U_1)\cos\theta_1 + (U_6 - U_2)\sin\theta_1] \tag{3.7}$$

and similarly for element 2 (Figure 3.3e):

$$f_3^{(2)} = -f_2^{(2)} = k^{(2)}\,[(U_5 - U_3)\cos\theta_2 + (U_6 - U_4)\sin\theta_2] \tag{3.8}$$

Note that, in writing Equations 3.7 and 3.8, we invoke the condition that the displacements of node 3 (U_5 and U_6) are the same for each element. To reiterate, this assumption is actually a requirement, since on a physical basis, the structure must remain connected at the joints after deformation. Displacement compatibility at the nodes is a fundamental requirement of the finite element method.

Substituting Equations 3.7 and 3.8 into the nodal equilibrium conditions (Equations 3.1–3.3) yields

$$-k^{(1)}[(U_5 - U_1)\cos\theta_1 + (U_6 - U_2)\sin\theta_1]\cos\theta_1 = F_1 \tag{3.9}$$

$$-k^{(1)}[(U_5 - U_1)\cos\theta_1 + (U_6 - U_2)\sin\theta_1]\sin\theta_1 = F_2 \tag{3.10}$$

$$-k^{(2)}[(U_5 - U_3)\cos\theta_2 + (U_6 - U_4)\sin\theta_2]\cos\theta_2 = F_3 \tag{3.11}$$

$$-k^{(2)}[(U_5 - U_3)\cos\theta_2 + (U_6 - U_4)\sin\theta_2]\sin\theta_2 = F_4 \tag{3.12}$$

$$\begin{aligned} &k^{(2)}[(U_5 - U_3)\cos\theta_2 + (U_6 - U_4)\sin\theta_2]\cos\theta_2 \\ &\quad + k^{(1)}[(U_5 - U_3)\cos\theta_1 + (U_6 - U_4)\sin\theta_1]\cos\theta_1 = F_5 \end{aligned} \tag{3.13}$$

$$\begin{aligned} &k^{(2)}[(U_5 - U_3)\cos\theta_2 + (U_6 - U_4)\sin\theta_2]\sin\theta_2 \\ &\quad + k^{(1)}[(U_5 - U_1)\cos\theta_1 + (U_6 - U_2)\sin\theta_1]\sin\theta_1 = F_6 \end{aligned} \tag{3.14}$$

Equations 3.9 through 3.14 are equivalent to the matrix form

$$\begin{bmatrix} k^{(1)}c^2\theta_1 & k^{(1)}s\theta_1 c\theta_1 & 0 & 0 & -k^{(1)}c^2\theta_1 & -k^{(1)}s\theta_1 c\theta_1 \\ k^{(1)}s\theta_1 c\theta_1 & k^{(1)}s^2\theta_1 & 0 & 0 & -k^{(1)}s\theta_1 c\theta_1 & -k^{(1)}s^2\theta_1 \\ 0 & 0 & k^{(2)}c^2\theta_2 & k^{(2)}s\theta_2 c\theta_2 & -k^{(2)}c^2\theta_2 & -k^{(2)}s\theta_2 c\theta_2 \\ 0 & 0 & k^{(2)}s\theta_2 c\theta_2 & k^{(2)}s^2\theta_2 & -k^{(2)}s\theta_2 c\theta_2 & -k^{(2)}s^2\theta_2 \\ -k^{(1)}c^2\theta_{12} & -k_1 s\theta_1 c\theta_1 & -k^{(2)}c^2\theta_2 & -k^{(2)}s\theta_2 c\theta_2 & \begin{matrix} k^{(1)}c^2\theta_1 + \\ k^{(2)}c^2\theta_2 \end{matrix} & \begin{matrix} k^{(1)}s\theta_1 c\theta_1 + \\ k^{(2)}s\theta_2 c\theta_2 \end{matrix} \\ -k_1 s\theta_1 c\theta_1 & -k^{(1)}s^2\theta_1 & -k^{(2)}s\theta_2 c\theta_2 & -k^{(2)}s^2\theta_2 & \begin{matrix} k^{(1)}s\theta_1 c\theta_1 + \\ k^{(2)}s\theta_2 c\theta_2 \end{matrix} & \begin{matrix} k^{(1)}s^2\theta_1 + \\ k^{(2)}s^2\theta_2 \end{matrix} \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} \tag{3.15}$$

The six algebraic equations represented by matrix Equation 3.15 express the complete set of equilibrium conditions for the two-element truss. Equation 3.15 is of the form

$$[K]\{U\} = \{F\} \tag{3.16}$$

where $[K]$ is the global stiffness matrix, $\{U\}$ is the vector of nodal displacements, and $\{F\}$ is the vector of applied nodal forces. We observe that the global stiffness matrix is a 6×6 symmetric matrix corresponding to six possible global displacements. Application of boundary conditions and solution of the equations are deferred at this time, pending further discussion.

3.3 ELEMENT TRANSFORMATION

Formulation of global finite element equations by direct application of equilibrium conditions, as in the previous section, proves to be quite cumbersome except for the very simplest of models. By writing the nodal equilibrium equations in the global coordinate system and introducing the displacement formulation, the procedure of the previous section implicitly transformed the individual element characteristics (the stiffness matrix) to the global system. A direct method for transforming the stiffness characteristics on an element-by-element basis is now developed in preparation for use in the direct assembly procedure of the following section.

Recalling the bar element equations expressed in the element frame as

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix}\begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix} \tag{3.17}$$

the present objective is to transform these equilibrium equations into the global coordinate system in the form

$$\left[K^{(e)}\right]\begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{Bmatrix} F_1^{(e)} \\ F_2^{(e)} \\ F_3^{(e)} \\ F_4^{(e)} \end{Bmatrix} \tag{3.18}$$

In Equation 3.18, $[K^{(e)}]$ represents the element stiffness matrix in the global coordinate system, the vector $\{F^{(e)}\}$ on the right-hand side contains the element nodal force components in the global frame, displacements $U_1^{(e)}$ and $U_3^{(e)}$ are parallel to the global X axis, while $U_2^{(e)}$ and $U_4^{(e)}$ are parallel to the global Y axis. The relation between the element axial displacements in the element coordinate system and the element displacements in global coordinates (Equation 3.4) is

$$u_1^{(e)} = U_1^{(e)} \cos\theta + U_2^{(e)} \sin\theta \tag{3.19}$$

$$u_2^{(e)} = U_3^{(e)} \cos\theta + U_4^{(e)} \sin\theta \tag{3.20}$$

which can be written in matrix form as

$$\begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix} \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} \tag{3.21}$$

where

$$[R] = \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix} \tag{3.22}$$

is the transformation matrix of element *axial* displacements to global displacements. (Again note that the element nodal displacements in the direction perpendicular to the element axis, v_1 and v_2, are not considered in the stiffness matrix development; these displacements come into play in dynamic analyses in Chapter 10.) Substituting Equation 3.22 into Equation 3.17 yields

$$\begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} \begin{bmatrix} \cos\theta & \sin\theta & 0 & 0 \\ 0 & 0 & \cos\theta & \sin\theta \end{bmatrix} \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix} \tag{3.23}$$

or

$$\begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix} \tag{3.24}$$

While we have transformed the equilibrium equations from element displacements to global displacements as the unknowns, the equations are still expressed in the element coordinate system. The first of Equation 3.23 is the equilibrium condition for element node 1 in the element coordinate system. If we multiply this equation by $\cos\theta$, we obtain the equilibrium equation for the node in the X direction of the global coordinate system. Similarly, multiplying by $\sin\theta$, the Y direction global equilibrium equation is obtained. Exactly the same procedure with the second equation expresses equilibrium of element node 2 in the global coordinate system. The same desired operations described are obtained if we premultiply both sides of Equation 3.24 by $[R]^T$, the transpose of the transformation matrix; that is,

$$[R]^T \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{bmatrix} \cos\theta & 0 \\ \sin\theta & 0 \\ 0 & \cos\theta \\ 0 & \sin\theta \end{bmatrix} \begin{Bmatrix} f_1^{(e)} \\ f_2^{(e)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(e)}\cos\theta \\ f_1^{(e)}\sin\theta \\ f_2^{(e)}\cos\theta \\ f_2^{(e)}\sin\theta \end{Bmatrix} \tag{3.25}$$

Clearly, the right-hand side of Equation 3.25 represents the components of the element forces in the global coordinate system, so we now have

$$[R]^T \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} = \begin{Bmatrix} F_1^{(e)} \\ F_2^{(e)} \\ F_3^{(e)} \\ F_4^{(e)} \end{Bmatrix} \tag{3.26}$$

Matrix Equation 3.26 represents the equilibrium equations for element nodes 1 and 2, expressed in the global coordinate system. Comparing this result with Equation 3.18, the element stiffness matrix in the global coordinate frame is seen to be given by

$$\left[K^{(e)}\right] = [R]^T \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \tag{3.27}$$

Introducing the notation $c = \cos\theta$, $s = \sin\theta$ and performing the matrix multiplications on the right-hand side of Equation 3.27 results in

$$\left[K^{(e)}\right] = k_e \begin{bmatrix} c^2 & sc & -c^2 & -sc \\ sc & s^2 & -sc & -s^2 \\ -c^2 & -sc & c^2 & sc \\ -sc & -s^2 & sc & s^2 \end{bmatrix} \tag{3.28}$$

where $k_e = AE/L$ is the characteristic axial stiffness of the element.

Examination of Equation 3.28 shows that the symmetry of the element stiffness matrix is preserved in the transformation to global coordinates. In addition, although not obvious by inspection, it can be shown that the determinant is zero, indicating that, after transformation, the stiffness matrix remains singular. This is to be expected, since as previously discussed, rigid body motion of the element is possible in the absence of specified constraints.

3.3.1 Direction Cosines

In practice, a finite element model is constructed by defining nodes at specified coordinate locations followed by definition of elements by specification of the nodes connected by each element. For the case at hand, nodes i and j are defined in global coordinates by (X_i, Y_i) and (X_j, Y_j). Using the nodal coordinates, element length is readily computed as

$$L = [(X_j - X_i)^2 + (Y_j - Y_i)^2]^{1/2} \tag{3.29}$$

and the unit vector directed from node i to node j is

$$\boldsymbol{\lambda} = \frac{1}{L}[(X_j - X_i)\mathbf{I} + (Y_j - Y_i)\mathbf{J}] = \cos\theta_X \mathbf{I} + \cos\theta_Y \mathbf{J} \tag{3.30}$$

where **I** and **J** are unit vectors in global coordinate directions X and Y, respectively. Recalling the definition of the scalar product of two vectors and referring again to Figure 3.4, the trigonometric values required to construct the element transformation matrix are also readily determined from the nodal coordinates as the *direction cosines* in Equation 3.30

$$\cos\theta = \cos\theta_X = \boldsymbol{\lambda}\cdot\mathbf{I} = \frac{X_j - X_i}{L} \tag{3.31}$$

$$\sin\theta = \cos\theta_Y = \boldsymbol{\lambda}\cdot\mathbf{J} = \frac{Y_j - Y_i}{L} \tag{3.32}$$

Thus, the element stiffness matrix of a bar element in global coordinates can be completely determined by specification of the nodal coordinates, the cross-sectional area of the element, and the modulus of elasticity of the element material.

3.4 DIRECT ASSEMBLY OF GLOBAL STIFFNESS MATRIX

Having addressed the procedure of transforming the element characteristics of the one-dimensional bar element into the global coordinate system of a two-dimensional structure, we now address a method of obtaining the global equilibrium equations via an element-by-element assembly procedure. The technique of directly assembling the global stiffness matrix for a finite element model of a truss is discussed in terms of the simple two-element system depicted in Figure 3.2. Assuming the geometry and material properties to be completely specified, the element stiffness matrix in the global frame can be formulated for each element using Equation 3.28 to obtain

$$\left[K^{(1)}\right] = \begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & k_{13}^{(1)} & k_{14}^{(1)} \\ k_{21}^{(1)} & k_{22}^{(1)} & k_{23}^{(1)} & k_{24}^{(1)} \\ k_{31}^{(1)} & k_{32}^{(1)} & k_{33}^{(1)} & k_{34}^{(1)} \\ k_{41}^{(1)} & k_{42}^{(1)} & k_{43}^{(1)} & k_{44}^{(1)} \end{bmatrix} \tag{3.33}$$

for element 1 and

$$\left[K^{(2)}\right] = \begin{bmatrix} k_{11}^{(2)} & k_{12}^{(2)} & k_{13}^{(2)} & k_{14}^{(2)} \\ k_{21}^{(2)} & k_{22}^{(2)} & k_{23}^{(2)} & k_{24}^{(2)} \\ k_{31}^{(2)} & k_{32}^{(2)} & k_{33}^{(2)} & k_{34}^{(2)} \\ k_{41}^{(2)} & k_{42}^{(2)} & k_{43}^{(2)} & k_{44}^{(2)} \end{bmatrix} \tag{3.34}$$

for element 2. The stiffness matrices given by Equations 3.33 and 3.34 contain 32 terms, which together will form the 6 × 6 system matrix containing 36 terms. To "assemble" the individual element stiffness matrices into the global stiffness matrix, it is necessary to observe the correspondence of individual element displacements to global displacements and allocate the associated element stiffness terms to the correct location in the global matrix. For element 1 of Figure 3.2, the element displacements correspond to global displacements per

$$\left\{U^{(1)}\right\} = \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} \Rightarrow \begin{Bmatrix} U_1 \\ U_2 \\ U_5 \\ U_6 \end{Bmatrix} \tag{3.35}$$

while for element 2

$$\left\{U^{(2)}\right\} = \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} \Rightarrow \begin{Bmatrix} U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} \tag{3.36}$$

Equations 3.35 and 3.36 are the connectivity relations for the truss and explicitly indicate how each element is connected in the structure. For example, Equation 3.35 clearly shows that element 1 is not associated with global displacements U_3 and U_4 (therefore, not connected to global node 2) and, hence, contributes no stiffness terms affecting those displacements. This means that element 1 has no effect on the third and fourth rows and columns of the global stiffness matrix. Similarly, element 2 contributes nothing to the first and second rows and columns.

Rather that write individual displacement relations, it is convenient to place all the element to global displacement data in a single table as shown in Table 3.1.

Table 3.1 Nodal Displacement Correspondence Table

Global Displacement	Element 1 Displacement	Element 2 Displacement
1	1	0
2	2	0
3	0	1
4	0	2
5	3	3
6	4	4

The first column contains the entire set of global displacements in numerical order. Each succeeding column represents an element and contains the number of the element displacement corresponding to the global displacement in each row. A zero entry indicates no connection, therefore no stiffness contribution. The individual terms in the global stiffness matrix are then obtained by allocating the element stiffness terms per the table as follows:

$$K_{11} = k_{11}^{(1)} + 0$$
$$K_{12} = k_{12}^{(1)} + 0$$
$$K_{13} = 0 + 0$$
$$K_{14} = 0 + 0$$
$$K_{15} = k_{13}^{(1)} + 0$$
$$K_{16} = k_{14}^{(1)} + 0$$
$$K_{22} = k_{22}^{(1)} + 0$$
$$K_{23} = 0 + 0$$
$$K_{24} = 0 + 0$$
$$K_{25} = k_{23}^{(1)} + 0$$
$$K_{26} = k_{24}^{(1)} + 0$$
$$K_{33} = 0 + k_{11}^{(2)}$$
$$K_{34} = 0 + k_{12}^{(2)}$$
$$K_{35} = 0 + k_{13}^{(2)}$$
$$K_{36} = 0 + k_{14}^{(2)}$$
$$K_{44} = 0 + k_{22}^{(2)}$$
$$K_{45} = 0 + k_{23}^{(2)}$$
$$K_{46} = 0 + k_{24}^{(2)}$$
$$K_{55} = k_{33}^{(1)} + k_{33}^{(2)}$$
$$K_{56} = k_{34}^{(1)} + k_{34}^{(2)}$$
$$K_{66} = k_{44}^{(1)} + k_{44}^{(2)}$$

where the known symmetry of the stiffness matrix has been implicitly used to avoid repetition. It is readily shown that the resulting global stiffness matrix is identical in every respect to that obtained in Section 3.2 via the equilibrium equations. This is the direct stiffness method; the global stiffness matrix is "assembled" by direct addition of the individual element stiffness terms per the nodal displacement correspondence table that defines element connectivity.

EXAMPLE 3.1

For the truss shown in Figure 3.2, $\theta_1 = \pi/4$, $\theta_2 = 0$, and the element properties are such that $k_1 = A_1E_1/L_1$, $k_2 = A_2E_2/L_2$. Transform the element stiffness matrix of each element into the global reference frame and assemble the global stiffness matrix.

■ **Solution**

For element 1, $\cos\theta_1 = \sin\theta_1 = \sqrt{2}/2$ and $c^2\theta_1 = s^2\theta_1 = c\theta_1 s\theta_1 = \dfrac{1}{2}$, so substitution into Equation 3.33 gives

$$[K^{(1)}] = \frac{k_1}{2}\begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix}$$

For element 2, $\cos\theta_2 = 1$, $\sin\theta_2 = 0$ which gives the transformed stiffness matrix as

$$[K^{(2)}] = k_2\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

Assembling the global stiffness matrix directly using Equations 3.35 and 3.36 gives

$$K_{11} = k_1/2$$
$$K_{12} = k_1/2$$
$$K_{13} = 0$$
$$K_{14} = 0$$
$$K_{15} = -k_1/2$$
$$K_{16} = -k_1/2$$
$$K_{22} = k_1/2$$
$$K_{23} = 0$$
$$K_{24} = 0$$
$$K_{25} = -k_1/2$$
$$K_{26} = -k_1/2$$
$$K_{33} = k_2$$
$$K_{34} = 0$$
$$K_{35} = -k_2$$
$$K_{36} = 0$$
$$K_{44} = 0$$
$$K_{45} = 0$$
$$K_{46} = 0$$

$$K_{55} = k_1/2 + k_2$$
$$K_{56} = k_1/2$$
$$K_{66} = k_1/2$$

The complete global stiffness matrix is then

$$[K] = \begin{bmatrix} k_1/2 & k_1/2 & 0 & 0 & -k_1/2 & -k_1/2 \\ k_1/2 & k_1/2 & 0 & 0 & -k_1/2 & -k_1/2 \\ 0 & 0 & k_2 & 0 & -k_2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -k_1/2 & -k_1/2 & -k_2 & 0 & k_1/2+k_2 & k_1/2 \\ -k_1/2 & -k_1/2 & 0 & 0 & k_1/2 & k_1/2 \end{bmatrix}$$

The previously described embodiment of the direct stiffness method is straightforward but cumbersome and inefficient in practice. The main problem inherent to the method lies in the fact that each term of the global stiffness matrix is computed sequentially and accomplishment of this sequential construction requires that each element be considered at each step. A technique that is much more efficient and well-suited to digital computer operations is now described. In the second method, the element stiffness matrix for each element is considered in sequence, and the element stiffness terms added to the global stiffness matrix per the nodal connectivity table. Thus, all terms of an individual element stiffness matrix are added to the global matrix, after which that element need not be considered further. To illustrate, we rewrite Equations 3.33 and 3.34 as

$$\left[K^{(1)}\right] = \begin{array}{cc} \begin{matrix} 1 & 2 & 5 & 6 \end{matrix} & \\ \begin{bmatrix} k_{11}^{(1)} & k_{12}^{(1)} & k_{13}^{(1)} & k_{14}^{(1)} \\ k_{21}^{(1)} & k_{22}^{(1)} & k_{23}^{(1)} & k_{24}^{(1)} \\ k_{31}^{(1)} & k_{32}^{(1)} & k_{33}^{(1)} & k_{34}^{(1)} \\ k_{41}^{(1)} & k_{42}^{(1)} & k_{43}^{(1)} & k_{44}^{(1)} \end{bmatrix} & \begin{matrix} 1 \\ 2 \\ 5 \\ 6 \end{matrix} \end{array} \tag{3.37}$$

$$\left[K^{(2)}\right] = \begin{array}{cc} \begin{matrix} 3 & 4 & 5 & 6 \end{matrix} & \\ \begin{bmatrix} k_{11}^{(2)} & k_{12}^{(2)} & k_{13}^{(2)} & k_{14}^{(2)} \\ k_{21}^{(2)} & k_{22}^{(2)} & k_{23}^{(2)} & k_{24}^{(2)} \\ k_{31}^{(2)} & k_{32}^{(2)} & k_{33}^{(2)} & k_{34}^{(2)} \\ k_{41}^{(2)} & k_{42}^{(2)} & k_{43}^{(2)} & k_{44}^{(2)} \end{bmatrix} & \begin{matrix} 3 \\ 4 \\ 5 \\ 6 \end{matrix} \end{array} \tag{3.38}$$

In this depiction of the stiffness matrices for the two individual elements, the numbers to the right of each row and above each column indicate the global displacement associated with the corresponding row and column of the element stiffness matrix. Thus, we combine the nodal displacement correspondence table with the individual element stiffness matrices. For the element matrices, each individual component is now labeled as associated with a specific row-column position of the global stiffness matrix and can be added directly to that location. For example, Equation 3.38 shows that the $k_{24}^{(2)}$ component of element 2 is to be added to global stiffness component K_{46} (and via symmetry K_{64}). Thus, we can take each element in turn and add the individual components of the element stiffness matrix to the proper locations in the global stiffness matrix.

The form of Equations 3.37 and 3.38 is convenient for illustrative purposes only. For actual computations, inclusion of the global displacement numbers within the element stiffness matrix is unwieldy. A streamlined technique suitable for computer application is described next. For a 2-D truss modeled by spar elements, the following conventions are adopted:

1. The global nodes at which each element is connected are denoted by i and j.
2. The origin of the element coordinate system is located at node i and the element x axis has a positive sense in the direction from node i to node j.
3. The global displacements at element nodes are U_{2i-1}, U_{2i}, U_{2j-1}, and U_{2j} as noted in Section 3.2.

Using these conventions, all the information required to define element connectivity and assemble the global stiffness matrix is embodied in an *element-node connectivity* table, which lists element numbers in sequence and shows the global node numbers i and j to which each element is connected. For the two-element truss of Figure 3.2, the required data are as shown in Table 3.2.

Using the nodal data of Table 3.2, we define, for each element, a 1×4 *element displacement location vector* as

$$\left[L^{(e)}\right] = [2i - 1 \quad 2i \quad 2j - 1 \quad 2j] \tag{3.39}$$

where each value is the global displacement number corresponding to element stiffness matrix rows and columns 1, 2, 3, 4 respectively. For the truss of Figure 3.2, the element displacement location vectors are

$$\left[L^{(1)}\right] = [1 \quad 2 \quad 5 \quad 6] \tag{3.40}$$

$$\left[L^{(2)}\right] = [3 \quad 4 \quad 5 \quad 6] \tag{3.41}$$

Before proceeding, let us note the quantity of information that can be obtained from simple-looking Table 3.2. With the geometry of the structure defined, the (X, Y) global coordinates of each node are specified. Using these data, the length of each element and the direction cosines of element orientation

Table 3.2 Element-Node Connectivity Table for Figure 3.2

	Node	
Element	***i***	***j***
1	1	3
2	2	3

are computed via Equations 3.29 and 3.30, respectively. Specification of the cross-sectional area A and modulus of elasticity E of each element allows computation of the element stiffness matrix in the global frame using Equation 3.28. Finally, the element stiffness matrix terms are added to the global stiffness matrix using the element displacement location vector.

In the context of the current example, the reader is to imagine a 6×6 array of mailboxes representing the global stiffness matrix, each of which is originally empty (i.e., the stiffness coefficient is zero). We then consider the stiffness matrix of an individual element in the (2-D) global reference frame. Per the location vector (addresses) for the element, the individual values of the element stiffness matrix are placed in the appropriate mailbox. In this fashion, each element is processed in sequence and its stiffness characteristics added to the global matrix. After all elements are processed, the array of mailboxes contains the global stiffness matrix.

3.5 BOUNDARY CONDITIONS, CONSTRAINT FORCES

Having obtained the global stiffness matrix via either the equilibrium equations or direct assembly, the system displacement equations for the example truss of Figure 3.2 are of the form

$$[K]\begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \\ U_5 \\ U_6 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \\ F_6 \end{Bmatrix} \tag{3.42}$$

As noted, the global stiffness matrix is a singular matrix; therefore, a unique solution to Equation 3.42 cannot be obtained directly. However, in developing these equations, we have not yet taken into account the constraints imposed on system displacements by the support conditions that must exist to preclude rigid body motion. In this example, we observe the displacement boundary conditions

$$U_1 = U_2 = U_3 = U_4 = 0 \tag{3.43}$$

leaving only U_5 and U_6 to be determined. Substituting the boundary condition values and expanding Equation 3.42 we have, formally,

$$\begin{aligned} K_{15}U_5 + K_{16}U_6 &= F_1 \\ K_{25}U_5 + K_{26}U_6 &= F_2 \\ K_{35}U_5 + K_{36}U_6 &= F_3 \\ K_{45}U_5 + K_{46}U_6 &= F_4 \\ K_{55}U_5 + K_{56}U_6 &= F_5 \\ K_{56}U_5 + K_{66}U_6 &= F_6 \end{aligned} \tag{3.44}$$

as the *reduced* system equations (this is the partitioned set of matrix equations, written explicitly for the active displacements). In this example, F_1, F_2, F_3, and F_4 are the components of the reaction forces at constrained nodes 1 and 2, while F_5 and F_6 are global components of applied external force at node 3. Given the external force components, the last two of Equations 3.44 can be explicitly solved for displacements U_5 and U_6. The values obtained for these two displacements are then substituted into the constraint equations (the first four of Equations 3.44) and the reaction force components computed.

A more general approach to application of boundary conditions and computation of reactions is as follows. Letting the subscript c denote constrained displacements and subscript a denote unconstrained (active) displacements, the system equations can be partitioned (Appendix A) to obtain

$$\begin{bmatrix} K_{cc} & K_{ca} \\ K_{ac} & K_{aa} \end{bmatrix} \begin{Bmatrix} U_c \\ U_a \end{Bmatrix} = \begin{Bmatrix} F_c \\ F_a \end{Bmatrix} \tag{3.45}$$

where the values of the constrained displacements U_c are known (but not necessarily zero), as are the applied external forces F_a. Thus, the unknown, active displacements are obtained via the lower partition as

$$[K_{ac}]\{U_c\} + [K_{aa}]\{U_a\} = \{F_a\} \tag{3.46a}$$

$$\{U_a\} = [K_{aa}]^{-1}(\{F_a\} - [K_{ac}]\{U_c\}) \tag{3.46b}$$

where we have assumed that the specified displacements $\{U_c\}$ are not necessarily zero, although that is usually the case in a truss structure. (Again, note that, for numerical efficiency, methods other than matrix inversion are applied to obtain the solutions formally represented by Equations 3.46.) Given the displacement solution of Equations 3.46, the reactions are obtained using the upper partition of matrix Equation 3.45 as

$$\{F_c\} = [K_{cc}]\{U_c\} + [K_{ca}]\{U_a\} \tag{3.47}$$

where $[K_{ca}] = [K_{ac}]^T$ by the symmetry property of the stiffness matrix.

3.6 ELEMENT STRAIN AND STRESS

The final computational step in finite element analysis of a truss structure is to utilize the global displacements obtained in the solution step to determine the strain and stress in each element of the truss. For an element connecting nodes i and j, the element nodal displacements *in the element coordinate system* are given by Equations 3.19 and 3.20 as

$$\begin{aligned} u_1^{(e)} &= U_1^{(e)} \cos\theta + U_2^{(e)} \sin\theta \\ u_2^{(e)} &= U_3^{(e)} \cos\theta + U_4^{(e)} \sin\theta \end{aligned} \tag{3.48}$$

and the element axial strain (utilizing Equation 2.29 and the discretization and interpolation functions of Equation 2.25) is then

$$\varepsilon^{(e)} = \frac{\mathrm{d}u^{(e)}(x)}{\mathrm{d}x} = \frac{\mathrm{d}^{(e)}}{\mathrm{d}x}[N_1(x) \quad N_2(x)] \begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix}$$
$$= \begin{bmatrix} \dfrac{-1}{L^{(e)}} & \dfrac{1}{L^{(e)}} \end{bmatrix} \begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \frac{u_2^{(e)} - u_1^{(e)}}{L^{(e)}} \tag{3.49}$$

where $L^{(e)}$ is element length. The element axial stress is then obtained via application of Hooke's law as

$$\sigma^{(e)} = E\varepsilon^{(e)} \tag{3.50}$$

Note, however, that the global solution does not give the element axial displacement directly. Rather, the element displacements are obtained from the global displacements via Equations 3.48. Recalling Equations 3.21 and 3.22, the element strain in terms of global system displacements is

$$\varepsilon^{(e)} = \frac{\mathrm{d}u^{(e)}(x)}{\mathrm{d}x} = \frac{\mathrm{d}}{\mathrm{d}x}[N_1(x) \quad N_2(x)][R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} \tag{3.51}$$

where $[R]$ is the element transformation matrix defined by Equation 3.22. The element stresses for the bar element in terms of global displacements are those given by

$$\sigma^{(e)} = E\varepsilon^{(e)} = E\frac{\mathrm{d}u^{(e)}(x)}{\mathrm{d}x} = E\frac{\mathrm{d}^{(e)}}{\mathrm{d}x}[N_1(x) \quad N_2(x)][R] \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \end{Bmatrix} \tag{3.52}$$

As the bar element is formulated here, a positive axial stress value indicates that the element is in tension and a negative value indicates compression per the usual convention. Note that the stress calculation indicated in Equation 3.52 must be performed on an element-by-element basis. If desired, the element forces can be obtained via Equation 3.23.

EXAMPLE 3.2

The two-element truss in Figure 3.5 is subjected to external loading as shown. Using the same node and element numbering as in Figure 3.2, determine the displacement components of node 3, the reaction force components at nodes 1 and 2, and the element displacements, stresses, and forces. The elements have modulus of elasticity $E_1 = E_2 = 10 \times 10^6$ lb/in.^{2} and cross-sectional areas $A_1 = A_2 = 1.5$ in.^{2}.

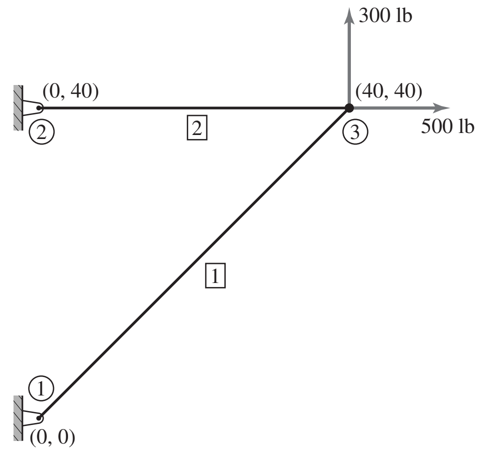


Figure 3.5 Two-element truss with external loading.

■ Solution

The nodal coordinates are such that $\theta_1 = \pi/4$ and $\theta_2 = 0$ and the element lengths are $L_1 = \sqrt{40^2 + 40^2} \approx 56.57$ in., $L_2 = 40$ in. The characteristic element stiffnesses are then

$$k_1 = \frac{A_1 E_1}{L_1} = \frac{1.5(10)(10^6)}{56.57} = 2.65(10^5) \text{ lb/in.}$$

$$k_2 = \frac{A_2 E_2}{L_2} = \frac{1.5(10)(10^6)}{40} = 3.75(10^5) \text{ lb/in.}$$

As the element orientation angles and numbering scheme are the same as in Example 3.1, we use the result of that example to write the global stiffness matrix as

$$[K] = \begin{bmatrix} 1.325 & 1.325 & 0 & 0 & -1.325 & -1.325 \\ 1.325 & 1.325 & 0 & 0 & -1.325 & -1.325 \\ 0 & 0 & 3.75 & 0 & -3.75 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -1.325 & -1.325 & -3.75 & 0 & 5.075 & 1.325 \\ -1.325 & -1.325 & 0 & 0 & 1.325 & 1.325 \end{bmatrix} 10^5 \text{ lb/in.}$$

Incorporating the displacement constraints $U_1 = U_2 = U_3 = U_4 = 0$, the global equilibrium equations are

$$10^5 \left[\begin{array}{cccc:cc} 1.325 & 1.325 & 0 & 0 & -1.325 & -1.325 \\ 1.325 & 1.325 & 0 & 0 & -1.325 & -1.325 \\ 0 & 0 & 3.75 & 0 & -3.75 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ \hdashline -1.325 & -1.325 & -3.75 & 0 & 5.075 & 1.325 \\ -1.325 & -1.325 & 0 & 0 & 1.325 & 1.325 \end{array}\right] \left\{\begin{array}{c} 0 \\ 0 \\ 0 \\ 0 \\ \hdashline U_5 \\ U_6 \end{array}\right\} = \left\{\begin{array}{c} F_1 \\ F_2 \\ F_3 \\ F_4 \\ \hdashline 500 \\ 300 \end{array}\right\}$$

and the dashed lines indicate the partitioning technique of Equation 3.45. Hence, the active displacements are governed by

$$10^5\begin{bmatrix} 5.075 & 1.325 \\ 1.325 & 1.325 \end{bmatrix}\begin{Bmatrix} U_5 \\ U_6 \end{Bmatrix} = \begin{Bmatrix} 500 \\ 300 \end{Bmatrix}$$

Simultaneous solution gives the displacements as

$$U_5 = 5.333 \times 10^{-4} \text{ in.} \quad \text{and} \quad U_6 = 1.731 \times 10^{-3} \text{ in.}$$

As all the constrained displacement values are zero, the reaction forces are obtained via Equation 3.47 as

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \{F_c\} = [K_{ca}]\{U_a\} = 10^5\begin{bmatrix} -1.325 & -1.325 \\ -1.325 & -1.325 \\ -3.75 & 0 \\ 0 & 0 \end{bmatrix}\begin{Bmatrix} 0.5333 \\ 1.731 \end{Bmatrix}10^{-3} = \begin{Bmatrix} -300 \\ -300 \\ -200 \\ 0 \end{Bmatrix} \text{lb}$$

and we note that the net force on the structure is zero, as required for equilibrium. A check of moments about any of the three nodes also shows that moment equilibrium is satisfied.

For element 1, the element displacements in the element coordinate system are

$$\begin{Bmatrix} u_1^{(1)} \\ u_2^{(1)} \end{Bmatrix} = \left[R^{(1)}\right]\begin{Bmatrix} U_1 \\ U_2 \\ U_5 \\ U_6 \end{Bmatrix} = \frac{\sqrt{2}}{2}\begin{bmatrix} 1 & 1 & 0 & 0 \\ 0 & 0 & 1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ 0 \\ 0.5333 \\ 1.731 \end{Bmatrix}10^{-3} = \begin{Bmatrix} 0 \\ 1.6 \end{Bmatrix}10^{-3} \text{ in.}$$

Element stress is computed using Equation 3.52:

$$\sigma^{(1)} = E_1\left[-\frac{1}{L_1} \quad \frac{1}{L_1}\right]\left[R^{(1)}\right]\begin{Bmatrix} U_1 \\ U_2 \\ U_5 \\ U_6 \end{Bmatrix}$$

Using the element displacements just computed, we have

$$\sigma^{(1)} = 10(10^6)\left[-\frac{1}{56.57} \quad \frac{1}{56.57}\right]\begin{Bmatrix} 0 \\ 1.6 \end{Bmatrix}10^{-3} \approx 283 \text{ lb/in.}^2$$

and the positive results indicate tensile stress.

The element nodal forces via Equation 3.23 are

$$\begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix}\begin{Bmatrix} u_1^{(1)} \\ u_2^{(1)} \end{Bmatrix} = 2.65(10^5)\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} 0 \\ 1.6 \end{Bmatrix}10^{-3}$$

$$= \begin{Bmatrix} -424 \\ 424 \end{Bmatrix} \text{lb}$$

and the algebraic signs of the element nodal forces also indicate tension.

For element 2, the same procedure in sequence gives

$$\begin{Bmatrix} u_1^{(2)} \\ u_2^{(2)} \end{Bmatrix} = \left[R^{(2)}\right] \begin{Bmatrix} U_1 \\ U_2 \\ U_5 \\ U_6 \end{Bmatrix} = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 0.5333 \\ 1.731 \end{Bmatrix} 10^{-3} = \begin{Bmatrix} 0 \\ 0.5333 \end{Bmatrix} 10^{-4} \text{ in.}$$

$$\sigma^{(2)} = 10(10^6)\left[-\frac{1}{40} \quad \frac{1}{40}\right] \begin{Bmatrix} 0 \\ 0.5333 \end{Bmatrix} 10^{-3} \approx 133 \text{ lb/in.}^2$$

$$\begin{Bmatrix} f_1^{(2)} \\ f_2^{(2)} \end{Bmatrix} = \begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1^{(2)} \\ u_2^{(2)} \end{Bmatrix} = 3.75(10^5) \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} 0 \\ 0.5333 \end{Bmatrix} 10^{-3} = \begin{Bmatrix} -200 \\ 200 \end{Bmatrix} \text{ lb}$$

also indicating tension.

The finite method is intended to be a general purpose procedure for analyzing problems for which the general solution is not known; however, it is informative in the examples of this chapter (since the bar element poses an exact formulation) to check the solutions in terms of axial stress computed simply as F/A for an axially loaded member. The reader is encouraged to compute the axial stress by the simple stress formula for each example to verify that the solutions via the stiffness-based finite element method are correct.

3.7 COMPREHENSIVE EXAMPLE

As a comprehensive example of two-dimensional truss analysis, the structure depicted in Figure 3.6a is analyzed to obtain displacements, reaction forces, strains, and stresses. While we do not include all computational details, the example illustrates the required steps, in sequence, for a finite element analysis.

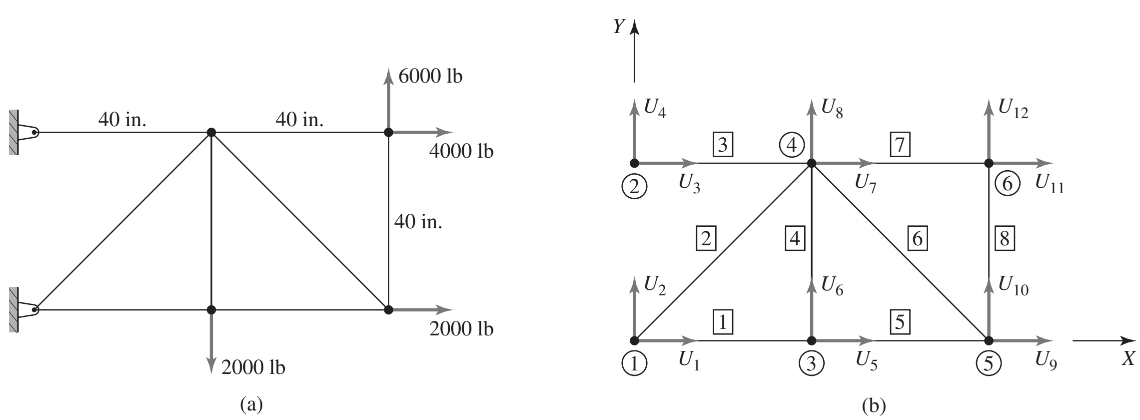


Figure 3.6
(a) For each element, $A = 1.5$ in.^{2}, $E = 10 \times 10^6$ psi. (b) Node, element, and global displacement notation.

Step 1. Specify the global coordinate system, assign node numbers, and define element connectivity, as shown in Figure 3.6b.

Step 2. Compute individual element stiffness values:

$$k^{(1)} = k^{(3)} = k^{(4)} = k^{(5)} = k^{(7)} = k^{(8)} = \frac{1.5(10^7)}{40} = 3.75(10^5) \text{ lb/in.}$$

$$k^{(2)} = k^{(6)} = \frac{1.5(10^7)}{40\sqrt{2}} = 2.65(10^5) \text{ lb/in.}$$

Step 3. Transform element stiffness matrices into the global coordinate system. Utilizing Equation 3.28 with

$$\theta_1 = \theta_3 = \theta_5 = \theta_7 = 0 \qquad \theta_4 = \theta_8 = \pi/2 \qquad \theta_2 = \pi/4 \qquad \theta_6 = 3\pi/4$$

we obtain

$$\left[K^{(1)}\right] = \left[K^{(3)}\right] = \left[K^{(5)}\right] = \left[K^{(7)}\right] = 3.75(10^5)\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

$$\left[K^{(4)}\right] = \left[K^{(8)}\right] = 3.75(10^5)\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

$$\left[K^{(2)}\right] = \frac{2.65(10^5)}{2}\begin{bmatrix} 1 & 1 & -1 & -1 \\ 1 & 1 & -1 & -1 \\ -1 & -1 & 1 & 1 \\ -1 & -1 & 1 & 1 \end{bmatrix}$$

$$\left[K^{(6)}\right] = \frac{2.65(10^5)}{2}\begin{bmatrix} 1 & -1 & -1 & 1 \\ -1 & 1 & 1 & -1 \\ -1 & 1 & 1 & -1 \\ 1 & -1 & -1 & 1 \end{bmatrix}$$

Step 4a. Construct the element-to-global displacement correspondence table. With reference to Figure 3.6c, the connectivity and displacement relations are shown in Table 3.3.

Step 4b. Alternatively and more efficiently, form the element-node connectivity table (Table 3.4), and the corresponding element global displacement location vector for each element is

$$L^{(1)} = [1 \quad 2 \quad 5 \quad 6]$$

$$L^{(2)} = [1 \quad 2 \quad 7 \quad 8]$$

$$L^{(3)} = [3 \quad 4 \quad 7 \quad 8]$$

$$L^{(4)} = [5 \quad 6 \quad 7 \quad 8]$$

Table 3.3 Connectivity and Displacement Relations

Global	Elem. 1	Elem. 2	Elem. 3	Elem. 4	Elem. 5	Elem. 6	Elem. 7	Elem. 8
1	1	1	0	0	0	0	0	0
2	2	2	0	0	0	0	0	0
3	0	0	1	0	0	0	0	0
4	0	0	2	0	0	0	0	0
5	3	0	0	1	1	0	0	0
6	4	0	0	2	2	0	0	0
7	0	3	3	3	0	3	1	0
8	0	4	4	4	0	4	2	0
9	0	0	0	0	3	1	0	1
10	0	0	0	0	4	2	0	2
11	0	0	0	0	0	0	3	3
12	0	0	0	0	0	0	4	4

Table 3.4 Element-Node Connectivity

	Node	
Element	i	j
1	1	3
2	1	4
3	2	4
4	3	4
5	3	5
6	5	4
7	4	6
8	5	6

$$L^{(5)} = [5 \quad 6 \quad 9 \quad 10]$$

$$L^{(6)} = [9 \quad 10 \quad 7 \quad 8]$$

$$L^{(7)} = [7 \quad 8 \quad 11 \quad 12]$$

$$L^{(8)} = [9 \quad 10 \quad 11 \quad 12]$$

Step 5. Assemble the global stiffness matrix per either Step 4a or 4b. The resulting components of the global stiffness matrix are

$$K_{11} = k_{11}^{(1)} + k_{11}^{(2)} = (3.75 + 2.65/2)10^5$$

$$K_{12} = k_{12}^{(1)} + k_{12}^{(2)} = (0 + 2.65/2)10^5$$

$$K_{13} = K_{14} = 0$$

$$K_{15} = k_{13}^{(1)} = -3.75(10^5)$$

$$K_{16} = k_{14}^{(1)} = 0$$

$$K_{17} = k_{13}^{(2)} = -(2.65/2)10^5$$

$$K_{18} = k_{14}^{(2)} = -(2.65/2)10^5$$
$$K_{19} = K_{1,10} = K_{1,11} = K_{1,12} = 0$$
$$K_{22} = k_{22}^{(1)} + k_{22}^{(2)} = 0 + (2.65/2)10^5$$
$$K_{23} = K_{24} = 0$$
$$K_{25} = k_{23}^{(1)} = 0$$
$$K_{26} = k_{24}^{(1)} = 0$$
$$K_{27} = k_{23}^{(2)} = -(2.65/2)10^5$$
$$K_{28} = k_{24}^{(2)} = -(2.65/2)10^5$$
$$K_{29} = K_{2,10} = K_{2,11} = K_{2,12} = 0$$
$$K_{33} = k_{11}^{(3)} = 3.75(10^5)$$
$$K_{34} = k_{12}^{(3)} = 0$$
$$K_{35} = K_{36} = 0$$
$$K_{37} = k_{13}^{(3)} = -3.75(10^5)$$
$$K_{38} = k_{14}^{(3)} = 0$$
$$K_{39} = K_{3,10} = K_{3,11} = K_{3,12} = 0$$
$$K_{44} = k_{22}^{(3)} = 0$$
$$K_{45} = K_{46} = 0$$
$$K_{47} = k_{23}^{(3)} = 0$$
$$K_{48} = k_{24}^{(3)} = 0$$
$$K_{49} = K_{4,10} = K_{4,11} = K_{4,12} = 0$$
$$K_{55} = k_{33}^{(1)} + k_{11}^{(4)} + k_{11}^{(5)} = (3.75 + 0 + 3.75)10^5$$
$$K_{56} = k_{34}^{(1)} + k_{12}^{(4)} + k_{12}^{(5)} = 0 + 0 + 0 = 0$$
$$K_{57} = k_{13}^{(4)} = 0$$
$$K_{58} = k_{14}^{(4)} = 0$$
$$K_{59} = k_{13}^{(5)} = -3.75(10^5)$$
$$K_{5,10} = k_{14}^{(5)} = 0$$
$$K_{5,11} = K_{5,12} = 0$$
$$K_{66} = k_{44}^{(2)} + k_{22}^{(4)} + k_{22}^{(5)} = (0 + 3.75 + 0)10^5$$
$$K_{67} = k_{23}^{(4)} = 0$$
$$K_{68} = k_{24}^{(4)} = -3.75(10)^5$$

$$
\begin{aligned}
K_{69} &= k_{23}^{(5)} = 0 \\
K_{6,10} &= k_{24}^{(5)} = 0 \\
K_{6,11} &= K_{6,12} = 0 \\
K_{77} &= k_{33}^{(2)} + k_{33}^{(3)} + k_{33}^{(4)} + k_{33}^{(6)} + k_{11}^{(7)} \\
&= (2.65/2 + 3.75 + 0 + 2.65/2 + 3.75)10^5 \\
K_{78} &= k_{34}^{(2)} + k_{34}^{(3)} + k_{34}^{(4)} + k_{34}^{(6)} + k_{12}^{(7)} \\
&= (2.65/2 + 0 + 0 - 2.65/2 + 0)10^5 = 0 \\
K_{79} &= k_{13}^{(6)} = -(2.65/2)10^5 \\
K_{7,10} &= k_{23}^{(6)} = (2.65/2)10^5 \\
K_{7,11} &= k_{13}^{(7)} = -3.75(10^5) \\
K_{7,12} &= k_{14}^{(7)} = 0 \\
K_{88} &= k_{44}^{(2)} + k_{44}^{(3)} + k_{44}^{(4)} + k_{44}^{(6)} + k_{22}^{(7)} \\
&= (2.65/2 + 0 + 3.75 + 2.65/2 + 0)10^5 \\
K_{89} &= k_{14}^{(6)} = (2.65/2)10^5 \\
K_{8,10} &= k_{24}^{(6)} = -(2.65/2)10^5 \\
K_{8,11} &= k_{23}^{(7)} = 0 \\
K_{8,12} &= k_{24}^{(7)} = 0 \\
K_{99} &= k_{33}^{(5)} + k_{11}^{(6)} + k_{11}^{(8)} = (3.75 + 2.65/2 + 0)10^5 \\
K_{9,10} &= k_{34}^{(5)} + k_{12}^{(6)} + k_{12}^{(8)} = (0 - 2.65/2 + 0)10^5 \\
K_{9,11} &= k_{13}^{(8)} = 0 \\
K_{9,12} &= k_{14}^{(8)} = 0 \\
K_{10,10} &= k_{44}^{(5)} + k_{22}^{(6)} + k_{22}^{(8)} = (0 + 2.65/2 + 3.75)10^5 \\
K_{10,11} &= k_{23}^{(8)} = 0 \\
K_{10,12} &= k_{24}^{(8)} = -3.75(10^5) \\
K_{11,11} &= k_{33}^{(7)} + k_{33}^{(8)} = (3.75 + 0)10^5 \\
K_{11,12} &= k_{34}^{(7)} + k_{34}^{(8)} = 0 + 0 \\
K_{12,12} &= k_{44}^{(7)} + k_{44}^{(8)} = (0 + 3.75)10^5
\end{aligned}
$$

Step 6. Apply the constraints as dictated by the boundary conditions. In this example, nodes 1 and 2 are fixed so the displacement constraints are

$$U_1 = U_2 = U_3 = U_4 = 0$$

Therefore, the first four equations in the 12 × 12 matrix system

$$[K]\{U\} = \{F\}$$

are constraint equations and can be removed from consideration since the applied displacements are all zero (if not zero, the constraints are considered as in Equation 3.46, in which case the nonzero constraints impose additional forces on the unconstrained displacements). The constraint forces cannot be obtained until the unconstrained displacements are computed. So, we effectively strike out the first four rows and columns of the global equations to obtain

$$[K_{aa}]\begin{Bmatrix} U_5 \\ U_6 \\ U_7 \\ U_8 \\ U_9 \\ U_{10} \\ U_{11} \\ U_{12} \end{Bmatrix} = \begin{Bmatrix} 0 \\ -2000 \\ 0 \\ 0 \\ 2000 \\ 0 \\ 4000 \\ 6000 \end{Bmatrix}$$

as the system of equations governing the "active" displacements.

Step 7. Solve the equations corresponding to the unconstrained displacements. For the current example, the equations are solved using a spreadsheet program, inverting the (relatively small) global stiffness matrix to obtain

$$\begin{Bmatrix} U_5 \\ U_6 \\ U_7 \\ U_8 \\ U_9 \\ U_{10} \\ U_{11} \\ U_{12} \end{Bmatrix} = \begin{Bmatrix} 0.02133 \\ 0.04085 \\ -0.01600 \\ 0.04619 \\ 0.04267 \\ 0.15014 \\ -0.00533 \\ 0.16614 \end{Bmatrix} \text{ in.}$$

Step 8. Back-substitute the displacement data into the constraint equations to compute reaction forces. Utilizing Equation 3.37, with $\{U_c\} = \{0\}$, we use the four equations previously ignored to compute the force components at nodes 1 and 2. The constraint equations are of the form

$$K_{i5}U_5 + K_{i6}U_6 + \cdots + K_{i,12}U_{12} = F_i \qquad i = 1, 4$$

and, on substitution of the computed displacements, yield

$$\begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{Bmatrix} -12{,}000 \\ -4{,}000 \\ 6{,}000 \\ 0 \end{Bmatrix} \text{ lb}$$

The reader is urged to utilize these reaction force components and check the equilibrium conditions of the structure.

Step 9. Compute strain and stress in each element. The major computational task completed in Step 7 provides the displacement components of each node in the global coordinate system. With this information and the known constrained displacements, the displacements of each element in its element coordinate system can be obtained; hence, the strain and stress in each element can be computed.

For element 2, for example, we have

$$u_1^{(2)} = U_1 \cos\theta_2 + U_2 \sin\theta_2 = 0$$

$$u_2^{(2)} = U_7 \cos\theta_2 + U_8 \sin\theta_2 = (-0.01600 + 0.04618)\sqrt{2}/2$$
$$= 0.02134$$

The axial strain in element 2 is then

$$\varepsilon^{(2)} = \frac{u_2^{(2)} - u_1^{(2)}}{L^{(2)}} = \frac{0.02133}{40\sqrt{2}} = 3.771(10^{-4})$$

and corresponding axial stress is

$$\sigma^{(2)} = E\varepsilon^{(2)} = 3771 \text{ psi}$$

The results for element 2 are presented as an example only. In finite element software, the results for each element are available and can be examined as desired by the user of the software (postprocessing).

Results for each of the eight elements are shown in Table 3.5; and per the usual sign convention, positive values indicate tensile stress while negative values correspond to compressive stress. In obtaining the computed results for this example, we used a spreadsheet program to invert the stiffness matrix, MATLAB to solve via matrix inversion, and a popular finite element software package. The solutions resulting from each procedure are identical.

Table 3.5 Results for the Eight Elements

Element	Strain	Stress, psi
1	$5.33(10^{-4})$	5333
2	$3.77(10^{-4})$	3771
3	$-4.0(10^{-4})$	−4000
4	$1.33(10^{-4})$	1333
5	$5.33(10^{-4})$	5333
6	$-5.67(10^{-4})$	−5657
7	$2.67(10^{-4})$	2667
8	$4.00(10^{-4})$	4000

3.8 THREE-DIMENSIONAL TRUSSES

Three-dimensional (3-D) trusses can also be modeled using the bar element, provided the connections between elements are such that only axial load is transmitted. Strictly, this requires that all connections be ball-and-socket joints. Even when the connection restriction is not precisely satisfied, analysis of a 3-D truss using bar elements is often of value in obtaining preliminary estimates of member stresses, which in context of design, is valuable in determining required structural properties. Referring to Figure 3.7 which depicts a one-dimensional bar element connected to nodes i and j in a 3-D global reference frame, the unit vector along the element axis (i.e., the element reference frame) expressed in the global system is

$$\boldsymbol{\lambda}^{(e)} = \frac{1}{L}[(X_j - X_i)\mathbf{I} + (Y_j - Y_i)\mathbf{J} + (Z_j - Z_i)\mathbf{K}] \tag{3.53}$$

or

$$\boldsymbol{\lambda}^{(e)} = \cos\theta_x\mathbf{I} + \cos\theta_y\mathbf{J} + \cos\theta_z\mathbf{K} \tag{3.54}$$

Thus, the element displacements are expressed in components in the 3-D global system as

$$u_1^{(e)} = U_1^{(e)}\cos\theta_x + U_2^{(e)}\cos\theta_y + U_3^{(e)}\cos\theta_z \tag{3.55}$$

$$u_2^{(e)} = U_4^{(e)}\cos\theta_x + U_5^{(e)}\cos\theta_y + U_6^{(e)}\cos\theta_z \tag{3.56}$$

Here, we use the notation that element displacements 1 and 4 are in the global X direction, displacements 2 and 5 are in the global Y direction, and element displacements 3 and 6 are in the global Z direction.

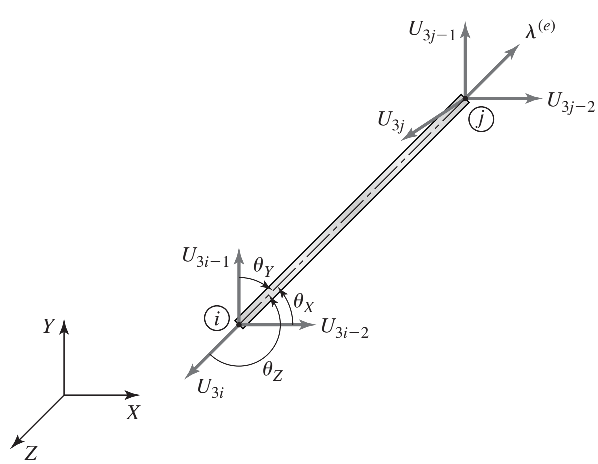


Figure 3.7 Bar element in a 3-D global coordinate system.

Analogous to Equation 3.21, Equations 3.55 and 3.56 can be expressed as

$$\begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = \begin{bmatrix} \cos\theta_x & \cos\theta_y & \cos\theta_z & 0 & 0 & 0 \\ 0 & 0 & 0 & \cos\theta_x & \cos\theta_y & \cos\theta_z \end{bmatrix} \begin{Bmatrix} U_1^{(e)} \\ U_2^{(e)} \\ U_3^{(e)} \\ U_4^{(e)} \\ U_5^{(e)} \\ U_6^{(e)} \end{Bmatrix} = [R]\{U^{(e)}\} \tag{3.57}$$

where $[R]$ is the transformation matrix mapping the one-dimensional element displacements into a three-dimensional global coordinate system. Following the identical procedure used for the 2-D case in Section 3.3, the element stiffness matrix in the element coordinate system is transformed into the 3-D global coordinates via

$$\left[K^{(e)}\right] = [R]^T \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} [R] \tag{3.58}$$

Substituting for the transformation matrix $[R]$ and performing the multiplication results in

$$\left[K^{(e)}\right] = k_e \begin{bmatrix} c_x^2 & c_x c_y & c_x c_z & -c_x^2 & -c_x c_y & -c_x c_z \\ c_x c_y & c_y^2 & c_y c_z & -c_x c_x & -c_y^2 & -c_y c_z \\ c_x c_z & c_y c_z & c_z^2 & -c_x c_z & -c_y c_z & -c_z^2 \\ -c_x^2 & -c_x c_x & -c_x c_z & c_x^2 & c_x c_y & c_x c_z \\ -c_x c_y & -c_y^2 & -c_y c_z & c_x c_y & c_y^2 & c_y c_z \\ -c_x c_z & -c_y c_z & -c_z^2 & c_x c_z & c_y c_z & c_z^2 \end{bmatrix} \tag{3.59}$$

as the 3-D global stiffness matrix for the one-dimensional bar element where

$$\begin{aligned} c_x &= \cos\theta_x \\ c_y &= \cos\theta_y \\ c_z &= \cos\theta_z \end{aligned} \tag{3.60}$$

Assembly of the global stiffness matrix (hence, the equilibrium equations), is identical to the procedure discussed for the two-dimensional case with the obvious exception that three displacements are to be accounted for at each node.

EXAMPLE 3.3

The three-member truss shown in Figure 3.8a is connected by ball-and-socket joints and fixed at nodes 1, 2, and 3. A 5000-lb force is applied at node 4 in the negative Y direction, as shown. Each of the three members is identical and exhibits a characteristic axial stiffness of $3(10^5)$ lb/in. Compute the displacement components of node 4 using a finite element model with bar elements.

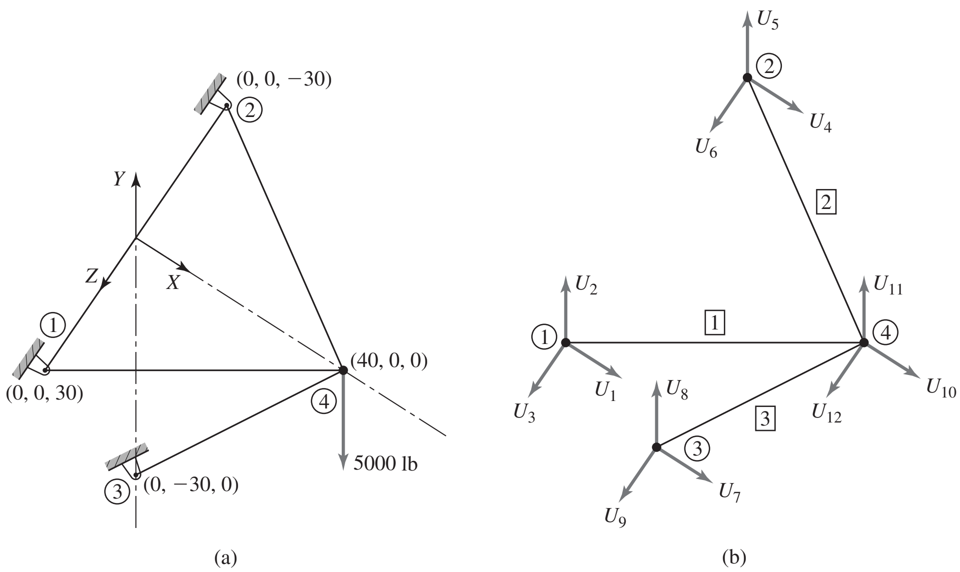


Figure 3.8
(a) A three-element, 3-D truss. (b) Numbering scheme.

■ Solution

First, note that the 3-D truss with four nodes has 12 possible displacements. However, since nodes 1–3 are fixed, nine of the possible displacements are known to be zero. Therefore, we need assemble only a portion of the system stiffness matrix to solve for the three unknown displacements. Utilizing the numbering scheme shown in Figure 3.8b and the element-to-global displacement correspondence table (Table 3.6), we need consider only the equations

$$\begin{bmatrix} K_{10,10} & K_{10,11} & K_{10,12} \\ K_{11,10} & K_{11,11} & K_{11,12} \\ K_{12,10} & K_{12,11} & K_{12,12} \end{bmatrix} \begin{Bmatrix} U_{10} \\ U_{11} \\ U_{12} \end{Bmatrix} = \begin{Bmatrix} 0 \\ -5000 \\ 0 \end{Bmatrix}$$

Prior to assembling the terms required in the system stiffness matrix, the individual element stiffness matrices must be transformed to the global coordinates as follows.

Element 1

$$\boldsymbol{\lambda}^{(1)} = \frac{1}{50}[(40-0)\mathbf{I} + (0-0)\mathbf{J} + (0-30)\mathbf{K}] = 0.8\mathbf{I} - 0.6\mathbf{K}$$

Hence, $c_x = 0.8$, $c_y = 0$, $c_z = -0.6$, and Equation 3.59 gives

$$\left[K^{(1)}\right] = 3(10^5)\begin{bmatrix} 0.64 & 0 & -0.48 & -0.64 & 0 & 0.48 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -0.48 & 0 & 0.36 & 0.48 & 0 & -0.36 \\ -0.64 & 0 & 0.48 & 0.64 & 0 & -0.48 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0.48 & -0 & -0.36 & -0.48 & 0 & 0.36 \end{bmatrix} \text{lb/ln.}$$

Table 3.6 Element-to-Global Displacement Correspondence

Global Displacement	Element 1	Element 2	Element 3
1	1	0	0
2	2	0	0
3	3	0	0
4	0	1	0
5	0	2	0
6	0	3	0
7	0	0	1
8	0	0	2
9	0	0	3
10	4	4	4
11	5	5	5
12	6	6	6

Element 2

$$\boldsymbol{\lambda}^{(2)} = \frac{1}{50}[(40-0)\mathbf{I} + (0-0)\mathbf{J} + (0-(-30))\mathbf{K}] = 0.8\mathbf{I} + 0.6\mathbf{K}$$

$$\left[K^{(2)}\right] = 3(10^5)\begin{bmatrix} 0.64 & 0 & 0.48 & -0.64 & 0 & -0.48 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0.48 & 0 & 0.36 & -0.48 & 0 & -0.36 \\ -0.64 & 0 & -0.48 & 0.64 & 0 & 0.48 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -0.48 & 0 & -0.36 & 0.48 & 0 & 0.36 \end{bmatrix} \text{lb/in.}$$

Element 3

$$\boldsymbol{\lambda}^{(3)} = \frac{1}{50}[(40-0)\mathbf{I} + (0-(-30))\mathbf{J} + (0-0)\mathbf{K}] = 0.8\mathbf{I} + 0.6\mathbf{J}$$

$$\left[K^{(3)}\right] = 3(10^5)\begin{bmatrix} 0.64 & 0.48 & 0 & -0.64 & -0.48 & 0 \\ 0.48 & 0.36 & 0 & -0.48 & -0.36 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ -0.64 & -0.48 & 0 & 0.64 & 0.48 & 0 \\ -0.48 & -0.36 & 0 & 0.48 & 0.36 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \text{lb/in.}$$

Referring to the last three rows of the displacement correspondence table, the required terms of the global stiffness matrix are assembled as follows:

$$K_{10,10} = k_{44}^{(1)} + k_{44}^{(2)} + k_{44}^{(3)} = 3(10^5)(0.64 + 0.64 + 0.64) = 5.76(10^5) \text{ lb/in.}$$

$$K_{10,11} = K_{11,10} = k_{45}^{(1)} + k_{45}^{(2)} + k_{45}^{(3)} = 3(10^5)(0 + 0 + 0.48) = 1.44(10^5) \text{ lb/in.}$$

$$K_{10,12} = K_{12,10} = k_{46}^{(1)} + k_{46}^{(2)} + k_{46}^{(3)} = 3(10^5)(-0.48 + 0.48 + 0) = 0 \text{ lb/in.}$$

$$K_{11,11} = k_{55}^{(1)} + k_{55}^{(2)} + k_{55}^{(3)} = 3(10^5)(0 + 0 + 0.36) = 1.08(10^5) \text{ lb/in.}$$

$$K_{11,12} = K_{12,11} = k_{56}^{(1)} + k_{56}^{(2)} + k_{56}^{(3)} = 3(10^5)(0 + 0 + 0) = 0 \text{ lb/in.}$$

$$K_{12,12} = k_{66}^{(1)} + k_{66}^{(2)} + k_{66}^{(3)} = 3(10^5)(0.36 + 0.36 + 0) = 2.16(10^5) \text{ lb/in.}$$

The system of equations to be solved for the displacements of node 4 are

$$10^5 \begin{bmatrix} 5.76 & 1.44 & 0 \\ 1.44 & 1.08 & 0 \\ 0 & 0 & 2.16 \end{bmatrix} \begin{Bmatrix} U_{10} \\ U_{11} \\ U_{12} \end{Bmatrix} = \begin{Bmatrix} 0 \\ -5000 \\ 0 \end{Bmatrix}$$

and simultaneous solution yields

$$U_{10} = 0.01736 \text{ in.}$$

$$U_{11} = -0.06944 \text{ in.}$$

$$U_{12} = 0$$

While the complete analysis is not conducted in the context of this example, the reaction forces, element strains, and element stresses would be determined by the same procedures followed in Section 3.7 for the two-dimensional case. It must be pointed out that the procedures required to obtain the individual element resultants are quite readily obtained by the matrix operations described here. Once the displacements have been calculated, the remaining (so-called) secondary variables (strain, stress, axial force) are readily computed using the matrices and displacement interpolation functions developed in the formulation of the original displacement problem.

3.9 SUMMARY

This chapter develops the complete procedure for performing a finite element analysis of a structure and illustrates it by several examples. Although only the simple axial element has been used, the procedure described is common to the finite element method for all element and analysis types, as will become clear in subsequent chapters. The direct stiffness method is by far the most straightforward technique for assembling the system matrices required for finite element analysis and is also very amenable to digital computer programming techniques.

REFERENCES

1. DaDeppo, D. *Introduction to Structural Mechanics and Analysis.* Upper Saddle River, NJ: Prentice-Hall, 1999.
2. Beer, F. P., and E. R. Johnston. *Vector Mechanics for Engineers, Statics and Dynamics,* 6th ed. New York: McGraw-Hill, 1997.

PROBLEMS

3.1 In the two-member truss shown in Figure 3.2, let $\theta_1 = 45°$, $\theta_2 = 15°$, and $F_5 = 5000$ lb, $F_6 = 3000$ lb.

a. Using only static force equilibrium equations, solve for the force in each member as well as the reaction force components.

b. Assuming each member has axial stiffness $k = 52000$ lb/in., compute the axial deflection of each member.

c. Using the results of part b, calculate the X and Y displacements of node 3.

3.2 Calculate the X and Y displacements of node 3 using the finite element approach and the data given in Problem 3.1. Also calculate the force in each element. How do your solutions compare to the results of Problem 3.1?

3.3 Verify Equation 3.28 by direct multiplication of the matrices.

3.4 Show that the transformed stiffness matrix for the bar element as given by Equation 3.28 is singular.

3.5 Each of the bar elements depicted in Figure P3.5 has a solid circular cross-section with diameter $d = 1.5$ in. The material is a low-carbon steel having modulus of elasticity $E = 30 \times 10^6$ psi. The nodal coordinates are given in a global (X, Y) coordinate system (in inches). Determine the element stiffness matrix of each element in the global system.

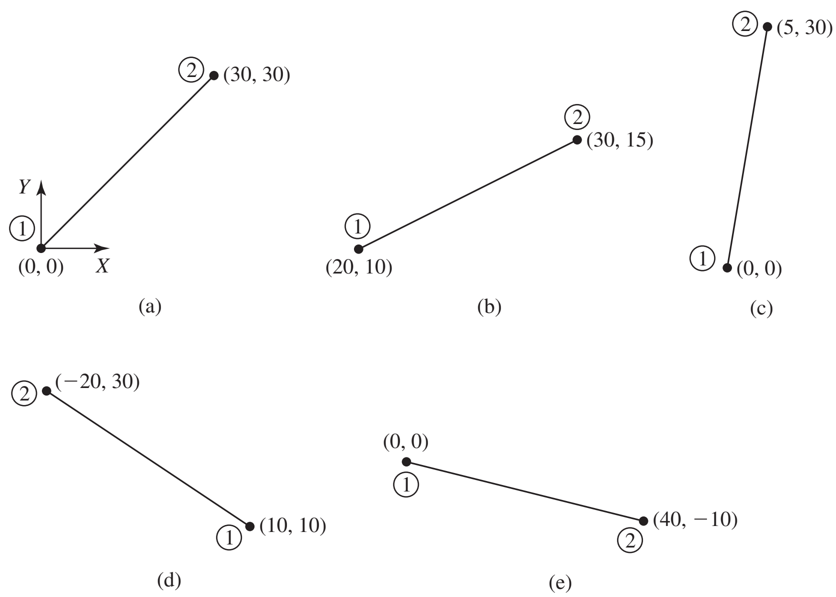


Figure P3.5

3.6 Repeat Problem 3.5 for the bar elements in Figure P3.6. For these elements, $d = 40$ mm, $E = 69$ GPa, and the nodal coordinates are in meters.

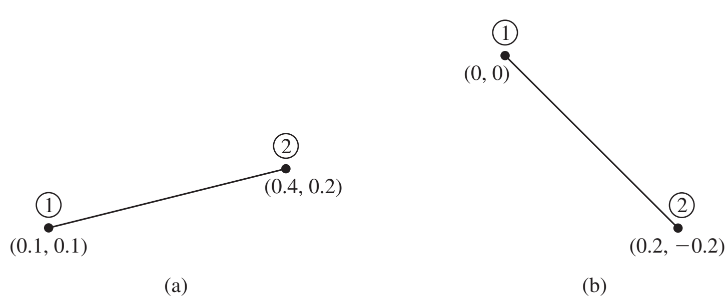


Figure P3.6

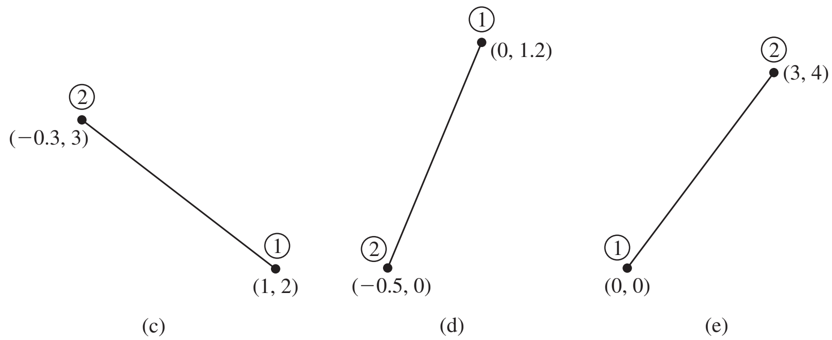
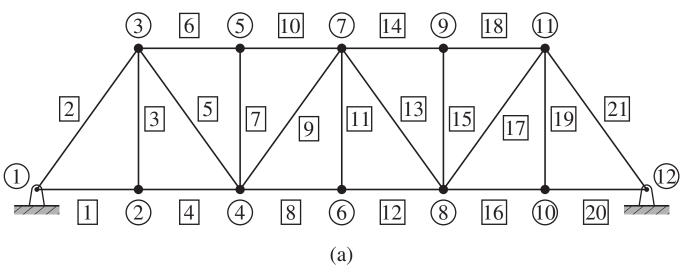


Figure P3.6 (*Continued*)

3.7 For each of the truss structures shown in Figure P3.7, construct an element-to-global displacement correspondence table in the form of Table 3.1.



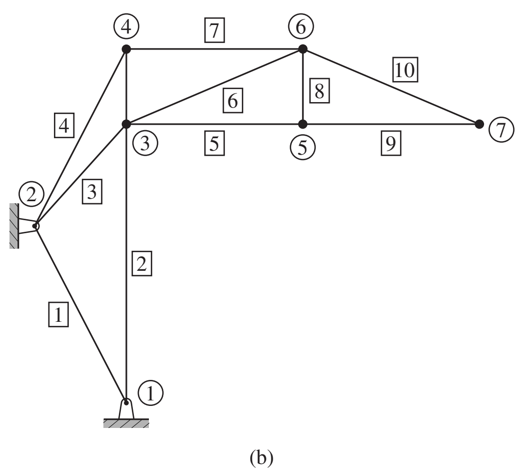


Figure P3.7

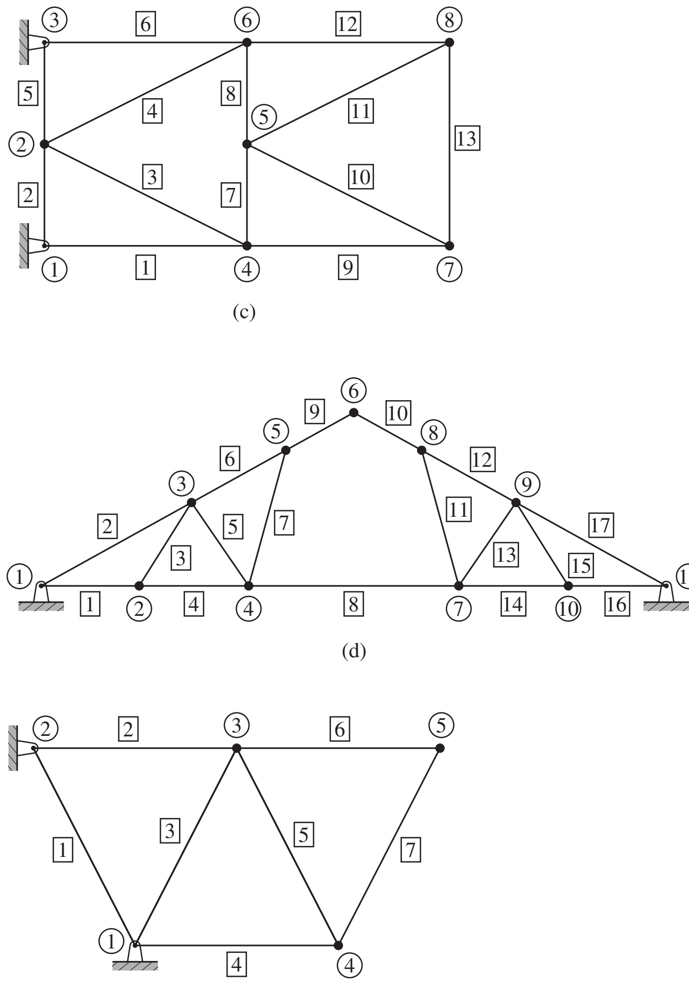


Figure P3.7 (*Continued*)

3.8 For each of the trusses of Figure P3.7, express the connectivity data for each element in the form of Equation 3.39.

3.9 For each element shown in Figure P3.9, the global displacements have been calculated as $U_1 = 0.05$ in., $U_2 = 0.02$ in., $U_3 = 0.075$ in., $U_4 = 0.09$ in. Using the finite element equations, calculate

a. Element axial displacements at each node.
b. Element strain.
c. Element stress.
d. Element nodal forces.

Do the calculated stress values agree with $\sigma = F/A$? Let $A = 0.75$ in.^{2}, $E = 10 \times 10^6$ psi, $L = 40$ in. for each case.

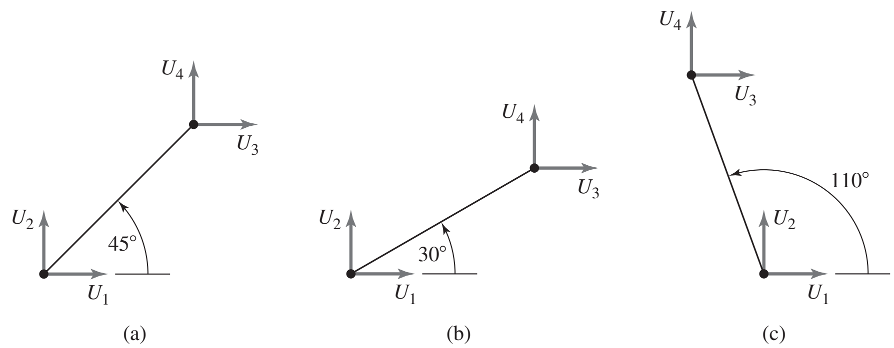


Figure P3.9

3.10 The plane truss shown in Figure P3.10 is subjected to a downward vertical load at node 2. Determine via the direct stiffness method the deflection of node 2 in the global coordinate system specified and the axial stress in each element. For both elements, $A = 0.5$ in.^{2}, $E = 30 \times 10^6$ psi.

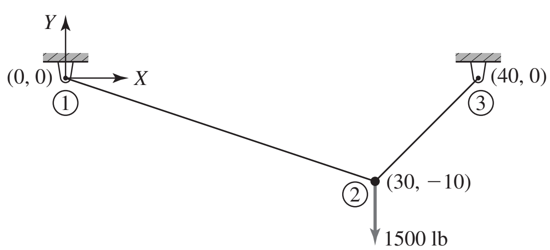


Figure P3.10

3.11 The plane truss shown in Figure P3.11 is composed of members having a square 15 mm × 15 mm cross section and modulus of elasticity $E = 69$ GPa.

a. Assemble the global stiffness matrix.
b. Compute the nodal displacements in the global coordinate system for the loads shown.
c. Compute the axial stress in each element.

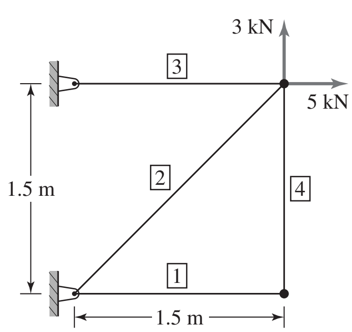
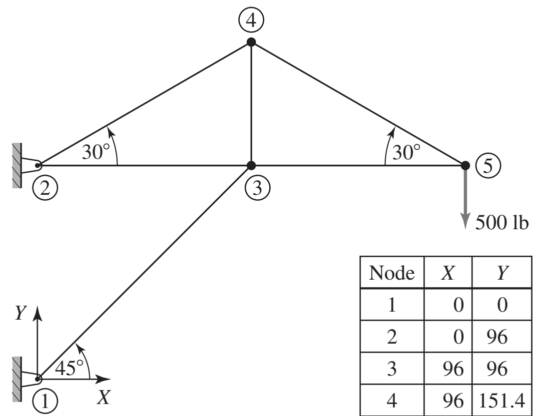


Figure P3.11

3.12 Repeat Problem 3.11 assuming elements 1 and 4 are removed.

3.13 The cantilever truss in Figure P3.13 was constructed by a builder to support a winch and cable system (not shown) to lift and lower construction materials. The truss members are nominal 2 × 4 southern yellow pine (actual dimensions 1.75 in. × 3.5 in.; $E = 2 \times 10^6$ psi). Using the direct stiffness method, calculate

a. The global displacement components of all unconstrained nodes.
b. Axial stress in each member.
c. Reaction forces at constrained nodes.
d. Check the equilibrium conditions.



Node	X	Y
1	0	0
2	0	96
3	96	96
4	96	151.4
5	192	96

(inches)

Figure P3.13

3.14 Figure P3.14 shows a two-member plane truss supported by a linearly elastic spring. The truss members are of a solid circular cross section having $d = 20$ mm and $E = 80$ GPa. The linear spring has stiffness constant 50 N/mm.

a. Assemble the system global stiffness matrix and calculate the global displacements of the unconstrained node.
b. Compute the reaction forces and check the equilibrium conditions.
c. Check the energy balance. Is the strain energy in balance with the mechanical work of the applied force?

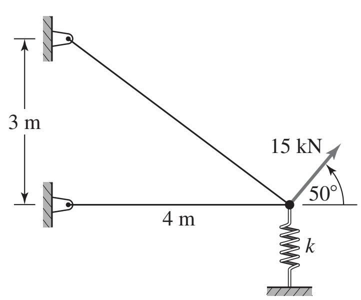


Figure P3.14

3.15 Repeat Problem 3.14 if the spring is removed.

3.16 Owing to a faulty support connection, node 1 in Problem 3.13 moves 0.5 in. horizontally to the left when the load is applied. Repeat the specified computations for this condition. Does the solution change? Why or why not?

3.17 Given the following system of algebraic equations

$$\begin{bmatrix} 10 & -10 & 0 & 0 \\ -10 & 20 & -10 & 0 \\ 0 & -10 & 20 & -10 \\ 0 & 0 & -10 & 10 \end{bmatrix} \begin{Bmatrix} x_1 \\ x_2 \\ x_3 \\ x_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix}$$

and the specified conditions

$$x_1 = 0 \qquad x_3 = 1.5 \qquad F_2 = 20 \qquad F_4 = 35$$

calculate x_2 and x_4. Do this by interchanging rows and columns such that x_1 and x_3 correspond to the first two rows and use the partitioned matrix approach of Equation 3.45.

3.18 Given the system

$$\begin{bmatrix} 50 & -50 & 0 & 0 \\ -50 & 100 & -50 & 0 \\ 0 & -50 & 75 & -25 \\ 0 & 0 & -25 & 25 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} 30 \\ F_2 \\ 40 \\ 40 \end{Bmatrix}$$

and the specified condition $U_2 = 0.5$, use the approach specified in Problem 3.17 to solve for U_1, U_3, U_4, and F_2.

3.19 For the truss shown in Figure P3.19, solve for the global displacement components of node 3 and the stress in each element. The elements have cross-sectional area $A = 1.0$ in.^{2} and modulus of elasticity 15×10^6 psi.

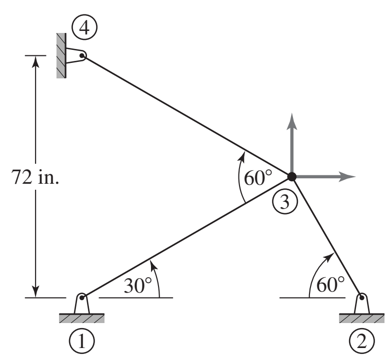


Figure P3.19

3.20 Each bar element shown in Figure P3.20 is part of a 3-D truss. The nodal coordinates (in inches) are specified in a global (X, Y, Z) coordinate system. Given $A = 2$ in.^{2} and $E = 30 \times 10^6$ psi, calculate the global stiffness matrix of each element.

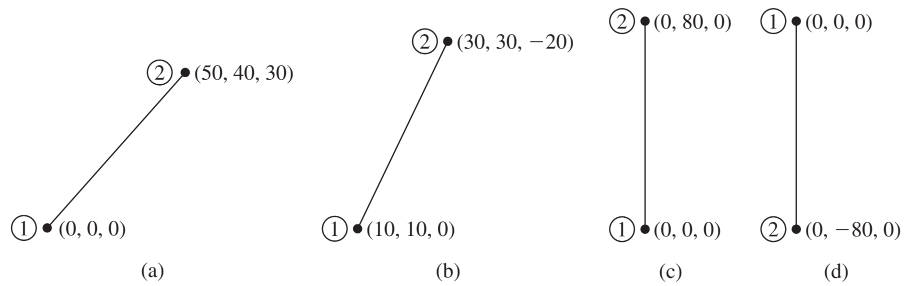


Figure P3.20

3.21 Verify Equation 3.59 via direct computation of the matrix product.

3.22 Show that the axial stress in a bar element in a 3-D truss is given by

$$\sigma = E\varepsilon = E\begin{bmatrix} \dfrac{dN_1}{dx} & \dfrac{dN_2}{dx} \end{bmatrix}\begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = E\begin{bmatrix} -\dfrac{1}{L} & \dfrac{1}{L} \end{bmatrix}[R]\{U^{(e)}\}$$

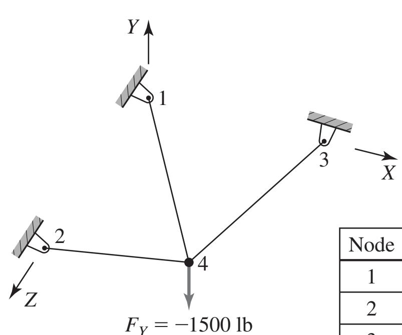
and note that the expression is the same as for the 2-D case.

3.23 Determine the axial stress and nodal forces for each bar element shown in Figure P3.20, given that node 1 is fixed and node 2 has global displacements $U_4 = U_5 = U_6 = 0.06$ in.

3.24 Use Equations 3.55 and 3.56 to express strain energy of a bar element in terms of the global displacements. Apply Castigliano's first theorem and show that the resulting global stiffness matrix is identical to that given by Equation 3.58.

3.25 Repeat Problem 3.24 using the principle of minimum potential energy.

3.26 Assemble the global stiffness matrix of the 3-D truss shown in Figure P3.26 and compute the displacement components of node 4. Also, compute the stress in each element.



Node	X	Y	Z
1	0	0	0
2	0	0	30
3	40	0	0
4	30	−20	25

Figure P3.26 Coordinates given in inches. For each element $E = 10 \times 10^6$ psi, $A = 1.5$ in.^{2}.