Chapter 1

Direct Method – Springs, Bars, and Truss Elements

An ability to predict the behavior of machines and engineering systems in general is of great importance at every stage of engineering processes, including design, manufacture, and operation. Such predictive methodologies are possible because engineers and scientists have made tremendous progress in understanding the physical behavior of materials and structures and have developed mathematical models, albeit approximate, in order to describe their physical behavior. Most often the mathematical models result in algebraic, differential, or integral equations or combinations thereof. Seldom can these equations be solved in closed form, and hence numerical methods are used to obtain solutions. The finite difference method is a classical method that provides approximate solutions to differential equations with reasonable accuracy. There are other methods of solving mathematical equations that are covered in traditional numerical methods courses[1].

The finite element method (FEM) is one of the numerical methods for solving differential equations. The FEM, originated in the area of structural mechanics, has been extended to other areas of solid mechanics and later to other fields such as heat transfer, fluid dynamics, and electromagnetism. In fact, FEM has been recognized as a powerful tool for solving partial differential equations and integro-differential equations, and it has become the numerical method of choice in many engineering and applied science areas. One of the reasons for FEM's popularity is that the method results in computer programs versatile in nature that can solve many practical problems with the least amount of training. Obviously, there is a danger in using computer programs without proper understanding of the theory behind them, and that is one of the reasons to have a thorough understanding of the theory behind the FEM.

The basic principle of FEM is to divide or *discretize* the system into a number of smaller elements called finite elements (FEs), to identify the degrees of freedom (DOFs) that describe its behavior, and then to write down the equations that describe the behavior of each element and its interaction with neighboring elements. The element-level equations are assembled to obtain global equations, often a linear system of equations, which are solved for the unknown DOFs. The phrase *finite element* refers to the fact that the elements are of a finite size as opposed to the infinitesimal or differential element considered in deriving the governing equations of the system. Another interpretation is that the FE equations deal with a finite number of DOFs as opposed to the infinite number of DOFs of a continuous system.

[1] Atkinson, K. E. 1978. *An Introduction to Numerical Analysis*. Wiley, New York.

Introduction to Finite Element Analysis and Design, Second Edition. Nam H. Kim, Bhavani V. Sankar, and Ashok V. Kumar.

Companion website: www.wiley.com/go/kim/finite_element_analysis_design

In general, solutions of practical engineering problems are quite complex, and they cannot be represented using simple mathematical expressions. An important concept of the FEM is that the solution is approximated using simple polynomials, often linear or quadratic, within each element. Since elements are connected throughout the system, the solution of the system is approximated using piecewise polynomials. Such approximation may contain errors when the size of an element is large. As the size of element reduces, however, the approximated solution will converge to the exact solution.

There are three methods that can be used to derive the FE equations of a problem: (a) direct method, (b) variational method, and (c) weighted residual method. The direct method provides a clear physical insight into the FEM and is preferred in the beginning stages of learning the principles. However, it is limited in its application in that it can be used to solve one-dimensional problems only. The variational method is akin to the methods of calculus of variations and is a powerful tool for deriving the FE equations. However, it requires the existence of a functional, whose minimization results in the solution of the differential equations. The Galerkin method is one of the popular weighted residual methods and is applicable to most problems. If a variational function exists for the problem, then the variational and Galerkin methods yield identical solutions.

In this chapter, we will illustrate the direct method of FE analysis using one-dimensional elements such as linear spring, uniaxial bar, and truss elements. The emphasis is on construction and solution of the finite element equations and interpretation of the results, rather than the rigorous development of the general principles of the FEM.

1.1 ILLUSTRATION OF THE DIRECT METHOD

Consider a system of rigid bodies connected by springs as shown in figure 1.1. The bodies move only in the horizontal direction. Furthermore, we consider only the static problem and hence the mass effects (inertia) will be ignored. External forces, F_2, F_3, and F_4, are applied on the rigid bodies as shown. The objectives are to determine the displacement of each body, forces in the springs, and support reactions.

We will introduce the principles involved in the FEM through this example. Notice that there is no need to discretize the system as it already consists of discrete elements, namely, the springs. The elements are connected at the nodes. In this case, the rigid bodies are the nodes. Of course, the two walls are also the nodes as they connect to the elements. Numbers inside the little circles mark the nodes. The system of connected elements is called the mesh and is best described using a connectivity table that defines which nodes an element is connected to as shown in table 1.1. Such a connectivity table is included in input files for finite element analysis software to describe the mesh.

Consider the free-body diagram of a typical element (e) as shown in figure 1.2. It has two nodes, nodes i and j. They will also be referred to as the first and second node or local node 1 (LN1) and local node 2 (LN2), respectively, as shown in the connectivity table. Assume a coordinate system going from left to right. The convention for first and second nodes is that $x_i < x_j$. The forces acting at the nodes are denoted by $f_i^{(e)}$ and $f_j^{(e)}$. In this notation, the subscripts denote the node numbers and the superscript the

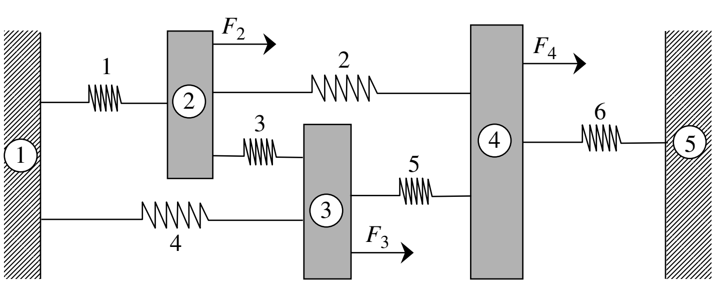


Figure 1.1 Rigid bodies connected by springs

Table 1.1 Connectivity table for figure 1.1

Element	LN1 (i)	LN2 (j)
1	1	2
2	2	4
3	2	3
4	1	3
5	3	4
6	4	5

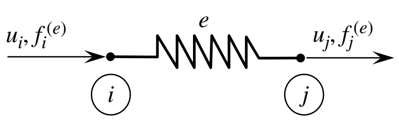


Figure 1.2 Spring element (e) connected by node i and node j

element number. This notation is adopted because multiple elements can be connected at a node, and each element may have different forces at the node. We will refer to them as *internal forces*. In figure 1.2, the forces are shown in the positive direction. The unknown displacements of nodes i and j are u_i and u_j, respectively. Note that there is no superscript for u, as the displacement is unique to the node denoted by the subscript. We would like to develop a relationship between the nodal displacements u_i and u_j and the internal forces $f_i^{(e)}$ and $f_j^{(e)}$.

The elongation of the spring is denoted by $\Delta^{(e)} = u_j - u_i$. Then the force of the spring is given by

$$P^{(e)} = k^{(e)}\Delta^{(e)} = k^{(e)}\left(u_j - u_i\right), \tag{1.1}$$

where $k^{(e)}$ is the spring rate or *stiffness* of element (e). In this text, the force in the spring, $P^{(e)}$, is referred to as *element force*. If $u_j > u_i$, then the spring is elongated, and the force in the spring is positive (tension). Otherwise, the spring is in compression. The spring element force is related to the internal force by

$$f_j^{(e)} = P^{(e)}. \tag{1.2}$$

Note that the sign of $f_i^{(e)}$ and $f_j^{(e)}$ is determined based on the direction that the force is applied, while the sign of $P^{(e)}$ is determined based on whether the element is in tension or compression. For equilibrium, the sum of the forces acting on element (e) must be equal to zero, i.e.,

$$f_i^{(e)} + f_j^{(e)} = 0 \quad \text{or} \quad f_i^{(e)} = -f_j^{(e)}. \tag{1.3}$$

Therefore, the two forces are equal, and they are applied in opposite directions. When $f_j^{(e)}$ is positive, the element is in tension, and thus, $P^{(e)}$ is positive.

From eqs. (1.1)–(1.3), we can obtain a relation between the internal forces and the displacements as

$$\begin{aligned} f_i^{(e)} &= k^{(e)}\left(u_i - u_j\right) \\ f_j^{(e)} &= k^{(e)}\left(-u_i + u_j\right). \end{aligned} \tag{1.4}$$

Equation (1.4) can be written in matrix forms as:

$$k^{(e)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i^{(e)} \\ f_j^{(e)} \end{Bmatrix}. \tag{1.5}$$

We also write eq. (1.5) in a shorthand notation as:

$$\left[\mathbf{k}^{(e)}\right]\begin{Bmatrix} u_i \\ u_j \end{Bmatrix} = \begin{Bmatrix} f_i^{(e)} \\ f_j^{(e)} \end{Bmatrix},$$

or,

$$\boxed{\left[\mathbf{k}^{(e)}\right]\left\{\mathbf{q}^{(e)}\right\} = \left\{\mathbf{f}^{(e)}\right\}}, \tag{1.6}$$

where $[\mathbf{k}^{(e)}]$ is the element stiffness matrix, $\{\mathbf{q}^{(e)}\}$ is the vector of DOFs associated with element (e), and $\{\mathbf{f}^{(e)}\}$ is the vector of internal forces. Sometimes we will omit the superscript (e) with the understanding that we are dealing with a generic element. Equation (1.6) is called the *element equilibrium equation*.

The element stiffness matrix $[\mathbf{k}^{(e)}]$ has the following properties:

1. It is square as it relates to the same number of forces as the displacements;
2. It is symmetric (a consequence of the Betti–Rayleigh Reciprocal theorem in solid and structural mechanics[2]);
3. It is singular, *i.e.*, its determinant is equal to zero, so it cannot be inverted; and
4. It is positive semidefinite.

Properties 3 and 4 are related to each other, and they have physical significance. Consider eq. (1.6). If the nodal displacements u_i and u_j of a spring element in a system are given, then it should be possible to predict the force $P^{(e)}$ in the spring from its change in length $\left(u_j - u_i\right)$, and hence the forces $\{\mathbf{f}^{(e)}\}$ acting at its nodes can be predicted. In fact, the internal forces can be computed by performing the matrix multiplication $[\mathbf{k}^{(e)}]\{\mathbf{q}^{(e)}\}$. On the other hand, if the two spring forces are given (they must have equal magnitudes but opposite directions), the nodal displacements cannot be determined uniquely, as a rigid body displacement (equal u_i and u_j) can be added without affecting the spring force. If $[\mathbf{k}^{(e)}]$ were to have an inverse, then it would have been possible to solve for $\left\{\mathbf{q}^{(e)}\right\} = \left[\mathbf{k}^{(e)}\right]^{-1}\left\{\mathbf{f}^{(e)}\right\}$ uniquely in violation of the physics. Property 4 has also a physical interpretation, which will be discussed in conjunction with energy methods.

In the next step, we develop a relationship between the internal forces $f_i^{(e)}$ and the known external forces F_i. For example, consider the free-body diagram of node 3 (or the rigid body in this case) in figure 1.1. The forces acting on the node are the external force F_3 and the internal forces from the springs connected to node 3 as shown in figure 1.3.

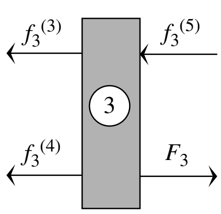


Figure 1.3 Free-body diagram of node 3 in the example shown in figure 1.1. The external force, F_3, and the forces, $f_3^{(e)}$, exerted by the springs attached to the node are shown. Note the forces $f_3^{(e)}$ act in the negative direction.

[2] Y. C. Fung. 1965. *Foundations of Solid Mechanics*. Prentice-Hall, Englewood Cliffs, NJ.

For equilibrium of the node, the sum of the forces acting on the node should be equal to zero:

$$F_i - \sum_{e=1}^{i_e} f_i^{(e)} = 0,$$

or,

$$F_i = \sum_{e=1}^{i_e} f_i^{(e)}, \quad i = 1, \ldots ND, \tag{1.7}$$

where i_e is the number of elements connected to node i, and ND is the total number of nodes in the model. Equation (1.7) is the equilibrium between externally applied forces at a node and internal forces from connected elements. If there is no externally applied force at a node, then the sum of internal forces at the node must be zero. Such equations can be written for each node including the boundary nodes, such as nodes 1 and 5 in figure 1.1. The internal forces $f_i^{(e)}$ in eq. (1.7) can be replaced by the unknown DOFs $\{\mathbf{q}\}$ by using eq. (1.6). For example, the force equilibrium for the springs in figure 1.1 can be written as

$$\begin{cases} F_1 = f_1^{(1)} + f_4^{(1)} = k^{(1)}(u_1 - u_2) + k^{(4)}(u_1 - u_3) \\ F_2 = f_2^{(1)} + f_2^{(3)} + f_2^{(2)} = k^{(1)}(u_2 - u_1) + k^{(3)}(u_2 - u_3) + k^{(2)}(u_2 - u_4) \\ F_3 = f_3^{(3)} + f_3^{(4)} + f_3^{(5)} = k^{(3)}(u_3 - u_2) + k^{(4)}(u_3 - u_1) + k^{(5)}(u_3 - u_4) \\ F_4 = f_4^{(2)} + f_4^{(5)} + f_4^{(6)} = k^{(2)}(u_4 - u_2) + k^{(5)}(u_4 - u_3) + k^{(6)}(u_4 - u_5) \\ F_5 = f_5^{(6)} = k^{(6)}(u_5 - u_4). \end{cases} \tag{1.8}$$

This will result in ND number of linear equations for the ND number of DOFs:

$$[\mathbf{K}_s] \begin{Bmatrix} u_1 \\ u_2 \\ \vdots \\ u_{ND} \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ \vdots \\ F_{ND} \end{Bmatrix}. \tag{1.9}$$

Or, in shorthand notation $[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}$ where $[\mathbf{K}_s]$ is the structural stiffness matrix, $\{\mathbf{Q}_s\}$ is the vector of displacements of all nodes in the model, and $\{\mathbf{F}_s\}$ is the vector of external forces, including the unknown reactions. The expanded form of eq. (1.9) is given in eq. (1.10) below:

$$\begin{bmatrix} k^{(1)} + k^{(4)} & -k^{(1)} & -k^{(4)} & 0 & 0 \\ -k^{(1)} & k^{(1)} + k^{(2)} + k^{(3)} & -k^{(3)} & -k^{(2)} & 0 \\ -k^{(4)} & -k^{(3)} & k^{(3)} + k^{(4)} + k^{(5)} & -k^{(5)} & 0 \\ 0 & -k^{(2)} & -k^{(5)} & k^{(2} + k^{(5)} + k^{(6)} & -k^{(6)} \\ 0 & 0 & 0 & -k^{(6)} & k^{(6)} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \\ F_5 \end{Bmatrix},$$

or,

$$[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}. \tag{1.10}$$

The properties of the structural stiffness matrix $[\mathbf{K}_s]$ are similar to that of the element stiffness matrix: square, symmetric, singular, and positive semi-definite. In addition, when nodes are numbered properly, $[\mathbf{K}_s]$ will be a banded matrix. It should be noted that when the boundary displacements in $\{\mathbf{Q}_s\}$ are known (usually equal to zero[3]), the corresponding forces in $\{\mathbf{F}_s\}$ are unknown reactions. In the present illustration, $u_1 = u_5 = 0$, and corresponding forces (reactions) F_1 and F_5 are unknown. It should also be noted that when displacements in $\{\mathbf{Q}_s\}$ are unknown, the corresponding forces in $\{\mathbf{F}_s\}$ should be known (either a given value or zero when no force is applied).

We will impose the boundary conditions as follows. First, we ignore the equations for which the RHS forces are unknown and strike out the corresponding rows in $[\mathbf{K}_s]$. This is called *striking the rows*. Then we eliminate the columns in $[\mathbf{K}_s]$ that are multiplied by the zero values of displacements of the boundary nodes. This is called *striking the columns*. It may be noted that if the n^{th} row is eliminated (struck), then the n^{th} column will also be eliminated (struck). This process results in a system of equations given by $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$, where $[\mathbf{K}]$ is the global stiffness matrix, $\{\mathbf{Q}\}$ is the vector of unknown DOFs, and $\{\mathbf{F}\}$ is the vector of known forces. The global stiffness matrix will be square, symmetric, and **positive definite** and hence nonsingular. Usually $[\mathbf{K}]$ will also be banded. In large systems, that is, in models with large numbers of DOFs, $[\mathbf{K}]$ will be a sparse matrix with a small proportion of nonzero numbers in a diagonal band.

After striking the rows and columns corresponding to zero DOFs (u_1 and u_5) in eq. (1.10), we obtain the global equations as follows:

$$\begin{bmatrix} k^{(1)}+k^{(2)}+k^{(3)} & -k^{(3)} & -k^{(2)} \\ -k^{(3)} & k^{(3)}+k^{(4)}+k^{(5)} & -k^{(5)} \\ -k^{(2)} & -k^{(5)} & k^{(2)}+k^{(5)}+k^{(6)} \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_2 \\ F_3 \\ F_4 \end{Bmatrix},$$

or,

$$[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}. \tag{1.11}$$

In principle, the solution can be obtained as $\{\mathbf{Q}\} = [\mathbf{K}]^{-1}\{\mathbf{F}\}$. Once the unknown DOFs are determined, the spring forces can be obtained using eq. (1.1). The support reactions can be obtained from either the nodal equilibrium equations (1.7) or the structural equations (1.10).

EXAMPLE 1.1 *Rigid body–spring system*

Find the displacements of the rigid bodies shown in figure 1.1. Assume that the only nonzero force is $F_3 = 1000$ N. Determine the element forces (tensile/compressive) in the springs. What are the reactions at the walls? Assume the bodies can undergo only translation in the horizontal direction. The spring constants (N/mm) are $k^{(1)} = 500$, $k^{(2)} = 400$, $k^{(3)} = 600$, $k^{(4)} = 200$, $k^{(5)} = 400$, and $k^{(6)} = 300$.

SOLUTION The element equilibrium equations are as follows:

$$\begin{gathered}
\begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix} = 500 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix}; \quad \begin{Bmatrix} f_2^{(2)} \\ f_4^{(2)} \end{Bmatrix} = 400 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_4 \end{Bmatrix} \\
\begin{Bmatrix} f_2^{(3)} \\ f_3^{(3)} \end{Bmatrix} = 600 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix}; \quad \begin{Bmatrix} f_1^{(4)} \\ f_3^{(4)} \end{Bmatrix} = 200 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_3 \end{Bmatrix} \\
\begin{Bmatrix} f_3^{(5)} \\ f_4^{(5)} \end{Bmatrix} = 400 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_3 \\ u_4 \end{Bmatrix}; \quad \begin{Bmatrix} f_4^{(6)} \\ f_5^{(6)} \end{Bmatrix} = 300 \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_4 \\ u_5 \end{Bmatrix}.
\end{gathered} \tag{1.12}$$

The nodal equilibrium equations are:

[3] Nonzero or prescribed DOFs will be dealt with in chapter 4.

$$
\begin{aligned}
f_1^{(1)} + f_1^{(4)} &= F_1 = R_1 \\
f_2^{(1)} + f_2^{(2)} + f_2^{(3)} &= F_2 = 0 \\
f_3^{(3)} + f_3^{(4)} + f_3^{(5)} &= F_3 = 1000 \\
f_4^{(2)} + f_4^{(5)} + f_4^{(6)} &= F_4 = 0 \\
f_5^{(6)} &= F_5 = R_5,
\end{aligned}
\tag{1.13}
$$

where R_1 and R_5 are unknown reaction forces at nodes 1 and 5, respectively. In the above equation, F_2 and F_4 are equal to zero because no external forces act on those nodes. Combining eqs. (1.12) and (1.13) we obtain the equation $[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}$,

$$
100\begin{bmatrix} 7 & -5 & -2 & 0 & 0 \\ -5 & 15 & -6 & -4 & 0 \\ -2 & -6 & 12 & -4 & 0 \\ 0 & -4 & -4 & 11 & -3 \\ 0 & 0 & 0 & -3 & 3 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} R_1 \\ 0 \\ 1000 \\ 0 \\ R_5 \end{Bmatrix}. \tag{1.14}
$$

After implementing the boundary conditions at nodes 1 and 5 (striking the rows and columns corresponding to zero displacements), we obtain the following global equations $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$:

$$
100\begin{bmatrix} 15 & -6 & -4 \\ -6 & 12 & -4 \\ -4 & -4 & 11 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 1000 \\ 0 \end{Bmatrix}.
$$

By inverting the global stiffness matrix, the unknown displacements can be obtained as: $u_2 = 0.854$ mm, $u_3 = 1.55$ mm, and $u_4 = 0.875$ mm.

The forces in the springs are computed using $P^{(e)} = k^{(e)}\left(u_j - u_i\right)$:

$$
\begin{aligned}
&P^{(1)} = 427\text{ N};\ P^{(2)} = 8.3\text{ N};\ P^{(3)} = 419\text{ N} \\
&P^{(4)} = 310\text{ N};\ P^{(5)} = -271\text{ N};\ P^{(6)} = -263\text{ N}.
\end{aligned}
$$

Wall reactions, R_1 and R_5, can be computed either from eq. (1.14) after substituting for the displacements, or from eqs. (1.12) and (1.13) as $R_1 = -737$ N; $R_5 = -263$ N. Both reactions are negative meaning that they act on the structure (the system) from right to left. ■

1.2 UNIAXIAL BAR ELEMENT

The FE analysis procedure for the spring–force system in the previous section can easily be extended to uniaxial bars. Plane and space trusses consist of uniaxial bars, and hence a detailed study of uniaxial bar finite element will provide the basis for analysis of trusses. Typical problems that can be solved using uniaxial bar elements are shown in figure 1.4. A uniaxial bar is a slender two-force member where the length is much larger than the cross-sectional dimensions. The bar can have varying cross-sectional area,$A(x)$, and consists of different materials, that is, varying Young's modulus, $E(x)$. Both concentrated forces F and distributed force $p(x)$ can be applied. The distributed forces can be applied over a portion of the bar. The forces F and $p(x)$ are considered positive if they act in the positive direction of the x-axis. Both ends of the bar can be fixed making it a statically indeterminate problem. Solving this problem by solving the differential equation of equilibrium could be difficult, if not impossible. However, this problem can be readily solved using FE analysis.

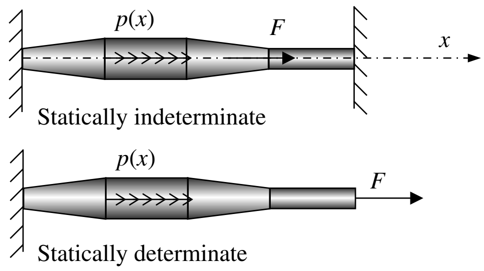


Figure 1.4 Typical one dimensional bar problems

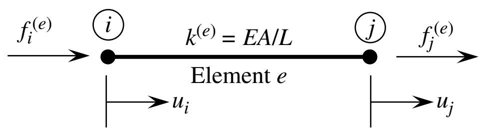


Figure 1.5 Uniaxial bar finite element

1.2.1 FE Formulation for Uniaxial Bar

The FE analysis procedures for the uniaxial bar are as follows:

1. Discretize the bar into a number of elements. The criteria for determining the size of the elements will become obvious after learning the properties of the element. It is assumed that each element has a constant axial rigidity,*EA*, throughout its length, although it may vary from element to element.
2. The elements are connected at nodes. Thus, more than one element can share a node. There will be nodes at points where the bar is supported.
3. External forces are applied only at the nodes, and they must be point forces (concentrated forces). If distributed forces are applied to the bar, they have to be approximated as point forces acting at nodes. At the bar boundary, if the displacement is specified, then the reaction is unknown. The reaction will be the external force acting on the boundary node. If a specified external force acts on the boundary, then the corresponding displacement is unknown. There will be no case when both displacement and force are unknown at a node.
4. The deformation of the bar is determined by the axial displacements of the nodes. That is, the nodal displacements are the DOFs in the FEM. Thus, the DOFs are u_1, u_2, u_3, …, u_N, where N is the total number of nodes.

The objective of the FE analysis is to determine: (i) unknown DOF (u_i); (ii) axial force resultant ($P^{(e)}$) in each element; and (iii) support reactions. Once the axial force resultant, $P^{(e)}$, is available, the element stress can easily be calculated by $\sigma = P^{(e)}/A^{(e)}$ where $A^{(e)}$ is the cross-section of the element.

We will use the *direct stiffness method* to derive the element stiffness matrix. Consider the free-body diagram of a typical element (e), as illustrated in figure 1.5. Forces and displacements are defined as positive when they are in the positive x direction. The element has two nodes, namely, i and j. Node i will be the first node and node j will be called the second node. The convention is that the line i–j will be in the positive direction of the x-axis. The displacements of the nodes are u_i and u_j. The element has a stiffness of $k^{(e)} = (EA/L)^{(e)}$ where EA is the axial rigidity, and L is the length of the element. It will be shown later that the stiffness $k^{(e)}$ plays exactly the same role as in the stiffness of a spring element in the previous section.

The forces acting at the two ends of the free body are $f_i^{(e)}$ and $f_j^{(e)}$. The superscript denotes the element number, and the subscripts denote the node numbers. The (lowercase) force f denotes the internal force as opposed to the (uppercase) external force F_i acting on the nodes. Since we do not know the direction of f, we will assume that all forces act in the positive coordinate direction. It should be noted that the nodal displacements do not need a superscript, as they are unique to the nodes. However, the internal force acting at a node may be different for different elements connected to the same node.

First, we will determine a relation between the f's and u's of the element (e). For equilibrium of the free-body diagram, we have

$$f_i^{(e)} + f_j^{(e)} = 0, \tag{1.15}$$

which means that the two forces acting on the two nodes of the element are equal and in opposite directions. Referring to figure 1.5, it is clear that when $f_j^{(e)} > 0$, the element is in tension, and when $f_j^{(e)} < 0$, the element is in compression.

From elementary mechanics of materials, the force is proportional to the elongation of the element. The elongation of the bar element is denoted by $\Delta^{(e)} = u_j - u_i$. Then, similar to the spring element, where $f = kx$, the force equilibrium of the one-dimensional bar element can be written, as

$$f_j^{(e)} = \left(\frac{AE}{L}\right)^{(e)} \left(u_j - u_i\right)$$

$$f_i^{(e)} = -f_j^{(e)} = \left(\frac{AE}{L}\right)^{(e)} \left(u_i - u_j\right),$$

where A, E, and L, respectively, are the area of the cross section, Young's modulus, and the length of the element. Using matrix notation, the above equations can be written as

$$\begin{Bmatrix} f_i^{(e)} \\ f_j^{(e)} \end{Bmatrix} = \left(\frac{AE}{L}\right)^{(e)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix}. \tag{1.16}$$

Equation (1.16) is called the *element equilibrium equation*, which relates the nodal forces of element (e) to the corresponding nodal displacements. Note that eq. (1.16) is similar to eq. (1.5) of the spring element if $k^{(e)} = (EA/L)^{(e)}$. Equation (1.16) for each element can be written in a compact form as

$$\left\{\mathbf{f}^{(e)}\right\} = \left[\mathbf{k}^{(e)}\right]\left\{\mathbf{q}^{(e)}\right\}, \quad e = 1, 2, \ldots, N_e, \tag{1.17}$$

where $[\mathbf{k}^{(e)}]$ is the element stiffness matrix of element (e), $\{\mathbf{q}^{(e)}\}$ is the vector of nodal displacements of the element, and N_e is the total number of elements in the model.

Note that the element stiffness matrix in eq. (1.16) is singular. The fact that the element stiffness matrix does not have an inverse has a physical significance. If the nodal displacements of an element are specified, then the element forces can be uniquely determined by performing the matrix multiplication in eq. (1.16). On the other hand, if the forces acting on the element are given, the nodal displacements cannot be uniquely determined because one can always translate the element by adding a rigid body displacement without affecting the forces acting on it. Thus, it is always necessary to remove the rigid body motion by fixing some displacements at nodes.

1.2.2 Nodal Equilibrium

Consider the free-body diagram of a typical node i. It is connected to, say, elements (e) and ($e + 1$). Then, the forces acting on the nodes are the external force F_i and reactions to the element forces as shown in

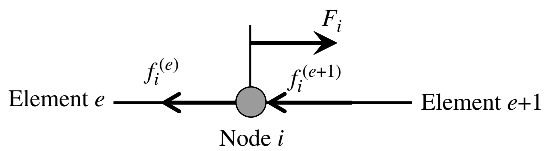


Figure 1.6 Force equilibrium at node i

figure 1.6. The internal forces are applied in the negative x direction because they are the reaction to the forces acting on the element. The sum of the forces acting on node i must be equal to zero:

$$F_i - f_i^{(e)} - f_i^{(e+1)} = 0,$$

or

$$f_i^{(e)} + f_i^{(e+1)} = F_i. \tag{1.18}$$

In general, the external force acting on a node is equal to sum of all the internal forces acting on different elements connected to the node, and eq. (1.18) can be generalized as

$$F_i = \sum_{e=1}^{i_e} f_i^{(e)}, \tag{1.19}$$

where i_e is the number of elements connected to node i, and the sum is carried out over all the elements connected to node i.

1.2.3 Assembly

The next step is to eliminate the internal forces from eq. (1.18) using eq. (1.17) in order to obtain a relation between the unknown displacements $\{\mathbf{Q}_s\}$ and known forces $\{\mathbf{F}_s\}$. This step results in a process called an *assembly* of the element stiffness matrices. We substitute for f's from eq. (1.17) into eq. (1.19) in order to find a relation between the nodal displacements and external forces. The force equilibrium in eq. (1.19) can be written for each DOF at each node yielding a relation between the external forces and displacements as

$$[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}. \tag{1.20}$$

Equation (1.20) is called the *structural matrix equation*. In the above equation, $[\mathbf{K}_s]$ is the structural stiffness matrix, which characterizes the load-deflection behavior of the entire structure; $\{\mathbf{Q}_s\}$ is the vector of all nodal displacements, known and unknown; and $\{\mathbf{F}_s\}$ is the vector of external forces acting at the nodes including the unknown reactions.

There is a systematic procedure by which the element stiffness matrices $[\mathbf{k}^{(e)}]$ can be assembled to obtain $[\mathbf{K}_s]$. We will assign a row address and column address for each entry in $[\mathbf{k}^{(e)}]$ and $[\mathbf{K}_s]$. The column address of a column is the DOF that the column multiplies with in the equilibrium equation. For example, the column addresses of the first and second column in $[\mathbf{k}^{(e)}]$ are u_i and u_j, respectively. The column addresses of columns 1, 2, 3,… in $[\mathbf{K}_s]$ are u_1, u_2, u_3,… respectively. The row addresses and column addresses are always symmetric. That is, the row address of the i^{th} row is same as the column address of the i^{th} column. Having determined the row and column addresses of $[\mathbf{k}^{(e)}]$ and $[\mathbf{K}_s]$, assembly of the element stiffness matrices can be done in a mechanical way. Each of the four entries (boxes) of an element stiffness matrix is transferred to the box in $[\mathbf{K}_s]$ with corresponding row and column addresses.

It is important to discuss the properties of the structural stiffness matrix $[\mathbf{K}_s]$. After assembly, the matrix $[\mathbf{K}_s]$ has the following properties:

1. It is square;
2. It is symmetric;

3. It is positive semi-definite;
4. Its determinant is equal to zero, and thus it does not have an inverse (it is singular);
5. The diagonal entries of the matrix are greater than or equal to zero.

For a given $\{\mathbf{Q}_s\}$, $\{\mathbf{F}_s\}$ can be determined uniquely; however, for a given $\{\mathbf{F}_s\}$, $\{\mathbf{Q}_s\}$ cannot be determined uniquely because an arbitrary rigid-body displacement can be added to $\{\mathbf{Q}_s\}$ without affecting $\{\mathbf{F}_s\}$.

1.2.4 Boundary Conditions

Before we solve eq. (1.20) we need to impose the displacement boundary conditions, that is, use the known nodal displacements in eq. (1.20). Mathematically, it means to make the global stiffness matrix positive definite so that the unknown displacements can be uniquely determined. Let us assume that the total size of $[\mathbf{K}_s]$ is $m \times m$. From the m equations, we will discard the equations for which we do not know the right-hand side (unknown reaction forces). This is called "striking-the-rows." The structural stiffness matrix becomes rectangular, as the number of equations is less than m. Now we delete the columns that will multiply into prescribed zero displacements in $\{\mathbf{Q}_s\}$. Usually, if the i^{th} row is deleted, then the i^{th} column will also be deleted. Thus, we will be deleting as many columns as we did for rows. This procedure is called "striking-the-columns." Now the stiffness matrix becomes square with size $n \times n$, where n is the number of unknown displacements. The resulting equations can be written as

$$[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}, \tag{1.21}$$

where $[\mathbf{K}]$ is the global stiffness matrix, $\{\mathbf{Q}\}$ are the unknown displacements, and $\{\mathbf{F}\}$ are the known external forces applied to nodes. Equation (1.21) is called the *global matrix equations*. In the structural matrix equations in eq. (1.20), the vector $\{\mathbf{Q}_s\}$ includes both known and unknown displacements. However, after applying boundary conditions, that is, striking the rows and striking the columns, the vector $\{\mathbf{Q}\}$ only includes unknown nodal displacements. For the same reason, the vector $\{\mathbf{F}\}$ only includes known external forces, not support reactions. The global stiffness matrix is always a positive definite matrix, which has an inverse. It is square symmetric and its diagonal elements are positive, that is, $K_{ii} > 0,\ i = 1, \ldots, n$. Thus, the displacements $\{\mathbf{Q}\}$ can be solved uniquely for a given set of nodal forces $\{\mathbf{F}\}$.

1.2.5 Calculation of Element Forces and Reaction Forces

Now that all the DOFs are known, the element force in element (e) can be determined using eq. (1.16). The axial force resultant $P^{(e)}$ in element (e) is given by

$$P^{(e)} = \left(\frac{AE}{L}\right)^{(e)} \Delta^{(e)} = \left(\frac{AE}{L}\right)^{(e)} \left(u_j - u_i\right). \tag{1.22}$$

The sign convention of axial force resultant is similar to that of stress. It is positive when the bar is in tension and negative when it is in compression. Another method of determining the axial-force resultant distribution along an element length is as follows. Consider the element equation (1.16). At the first node or node i, the axial force is given by $P_i = -f_i$. That is, if f_i acts in the positive direction, that end is under compression. If f_i is in the negative direction, the element is under tension. On the other hand, the opposite is true at the second node, node j. In that case, $P_j = +f_j$. Then, we can modify eq. (1.16) as

$$\begin{Bmatrix} -P_i^{(e)} \\ +P_j^{(e)} \end{Bmatrix} = \left(\frac{AE}{L}\right)^{(e)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} u_i \\ u_j \end{Bmatrix}. \tag{1.23}$$

It happens that $P_i^{(e)} = P_j^{(e)}$, and hence we use a single variable $P^{(e)}$ to denote the axial force in an element as shown in (1.22).

It is important to realize that according to the convention used in structural mechanics, the reactions are forces acting on the structure exerted by the supports. There are two methods of determining the support reactions. The straightforward method is to use eq. (1.20) to determine the unknown $\{\mathbf{F}_s\}$. However, in some FE programs, the structural stiffness matrix $[\mathbf{K}_s]$ is never assembled. The striking of rows and columns is performed at element level, and the global stiffness matrix $[\mathbf{K}]$ is assembled directly. In such situations, eq. (1.19) is used to compute the reactions. For example, the reaction at the i^{th} node is obtained by computing the internal forces in the elements connected to node i and summing all the internal forces.

EXAMPLE 1.2 *Clamped-clamped uniaxial bar*

Use FEM to determine the axial force P in each portion, AB and BC, of the uniaxial bar shown in figure 1.7. What are the support reactions at A and C? Young's modulus is $E = 100$ GPa; the areas of the cross sections of the two portions AB and BC are, respectively, 1×10^{-4} m^2 and 2×10^{-4} m^2 and $F = 10{,}000$ N. The force F is applied at the cross section at B.

SOLUTION: Since the applied force is a concentrated or point force, it is sufficient to use two elements, AB and BC. The nodes A, B, and C, respectively, will be nodes 1, 2, and 3.

Using eq. (1.16), the element stiffness matrices for two elements are first calculated by

$$\left[\mathbf{k}^{(1)}\right] = \frac{10^{11} \times 10^{-4}}{0.25}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^7 \overset{\begin{matrix} u_1 & u_2 \end{matrix}}{\begin{bmatrix} 4 & -4 \\ -4 & 4 \end{bmatrix}} \begin{matrix} u_1 \\ u_2 \end{matrix},$$

$$\left[\mathbf{k}^{(2)}\right] = \frac{10^{11} \times 2 \times 10^{-4}}{0.4}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} = 10^7 \overset{\begin{matrix} u_2 & u_3 \end{matrix}}{\begin{bmatrix} 5 & -5 \\ -5 & 5 \end{bmatrix}} \begin{matrix} u_2 \\ u_3 \end{matrix}.$$

Note that the row addresses are written against each row in the element stiffness matrices, and column addresses are shown above each column. Using eqs. (1.19) and (1.20), the two elements are assembled to produce the structural equilibrium equations:

$$10^7 \overset{\begin{matrix} u_1 & u_2 & u_3 \end{matrix}}{\begin{bmatrix} 4 & -4 & 0 \\ -4 & 4+5 & -5 \\ 0 & -5 & 5 \end{bmatrix}} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 10{,}000 \\ F_3 \end{Bmatrix}. \tag{1.24}$$

Note that nodes 1 and 3 are fixed and have unknown reaction forces. After deleting the rows and columns corresponding to the fixed DOFs (u_1 and u_3), we obtain $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$:

$$10^7[9]\{u_2\} = \{10{,}000\} \quad \Rightarrow \quad u_2 = 1.111 \times 10^{-4}\,\text{m}.$$

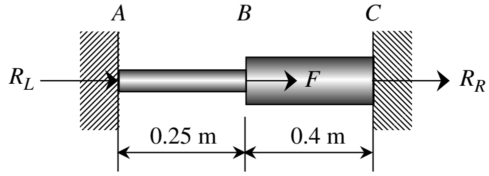


Figure 1.7 Two clamped uniaxial bars

Note that the final equation turns out to be a scalar equation because there is only one free DOF. By collecting all DOFs, the vector of nodal displacements can be obtained as: $\{\mathbf{Q}_s\}^{\mathrm{T}} = \{u_1, u_2, u_3\} = \{0,\ 1.111 \times 10^{-4},\ 0\}$. After solving for the unknown nodal displacements, the axial forces of the elements can be computed using $P = (AE/L)(u_j - u_i)$, as

$$P^{(1)} = 4 \times 10^7 (u_2 - u_1) = 4{,}444\mathrm{N}$$
$$P^{(2)} = 5 \times 10^7 (u_3 - u_2) = -5{,}556\mathrm{N}.$$

Note that the first element is under tension, while the second is under compressive force.

The reaction forces can be calculated from the first and third rows in eq. (1.24) using the calculated nodal DOFs, as

$$R_L = F_1 = -4 \times 10^7 u_2 = -4{,}444\mathrm{N},$$
$$R_R = F_2 = -5 \times 10^7 u_2 = -5{,}556\mathrm{N}.$$

Alternatively, from the equilibrium between internal and external forces [eq. (1.19)], the two reaction forces can be calculated using the internal forces, as

$$R_L = -P^{(1)} = -4{,}444N,$$
$$R_R = +P^{(2)} = -5{,}556N.$$

Note that both reaction forces are in the negative x direction, and the sum of reactions is the same as the external force at node 2 with the opposite sign. ■

EXAMPLE 1.3 *Three uniaxial bar elements*

Consider an assembly of three two-force members as shown in figure 1.8. Motion is restricted to one dimension along the x-axis. Determine the displacement of the rigid member, element forces, and reaction forces from the wall. Assume $k^{(1)} = 50$ N/cm, $k^{(2)} = 30$ N/cm, $k^{(3)} = 70$ N/cm, and $F_1 = 40$ N.

SOLUTION The assembly consists of three elements and four nodes. Figure 1.9 illustrates the free-body diagram of the system with node and element numbers.

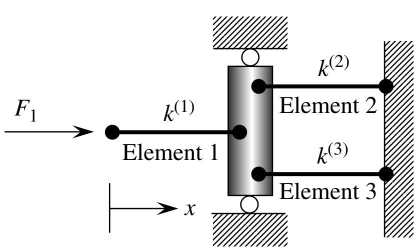


Figure 1.8 One-dimensional structure with three uniaxial bar elements

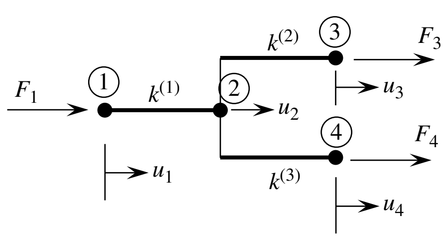


Figure 1.9 Finite element model

Write down the stiffness matrix of each element along with the row addresses. From now on, we will not show the column addresses over the stiffness matrices.

Element 1: $\left[\mathbf{k}^{(1)}\right] = k^{(1)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{matrix} u_1 \\ u_2 \end{matrix}.$

Element 2: $\left[\mathbf{k}^{(2)}\right] = k^{(2)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{matrix} u_2 \\ u_3 \end{matrix}.$

Element 3: $\left[\mathbf{k}^{(3)}\right] = k^{(3)} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{matrix} u_2 \\ u_4 \end{matrix}.$

After assembling the element stiffness matrices, we obtain the following structural stiffness matrix:

$$[\mathbf{K}_s] = \underbrace{\begin{bmatrix} k^{(1)} & -k^{(1)} & 0 & 0 \\ -k^{(1)} & \left(k^{(1)} + k^{(2)} + k^{(3)}\right) & -k^{(2)} & -k^{(3)} \\ 0 & -k^{(2)} & k^{(2)} & 0 \\ 0 & -k^{(3)} & 0 & k^{(3)} \end{bmatrix}}_{\text{Structural Stiffness Matrix}} \begin{matrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{matrix}.$$

The equation $[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}$ takes the form:

$$\begin{bmatrix} k^{(1)} & -k^{(1)} & 0 & 0 \\ -k^{(1)} & \left(k^{(1)} + k^{(2)} + k^{(3)}\right) & -k^{(2)} & -k^{(3)} \\ 0 & -k^{(2)} & k^{(2)} & 0 \\ 0 & -k^{(3)} & 0 & k^{(3)} \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix}. \tag{1.25}$$

The next step is to substitute boundary conditions and solve for unknown displacements. At all nodes, either the externally applied load or the displacement is specified. Substituting for the stiffnesses $k^{(1)}$, $k^{(2)}$, and $k^{(3)}$, $F_1 = 40$ N and $F_2 = 0$, and $u_3 = u_4 = 0$ in eq. (1.25), we obtain

$$\begin{Bmatrix} F_1 = 40 \\ F_2 = 0 \\ F_3 = R_3 \\ F_4 = R_4 \end{Bmatrix} = \begin{bmatrix} 50 & -50 & 0 & 0 \\ -50 & (50 + 30 + 70) & -30 & -70 \\ 0 & -30 & 30 & 0 \\ 0 & -70 & 0 & 70 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 = 0 \\ u_4 = 0 \end{Bmatrix}. \tag{1.26}$$

Next, we delete the rows and columns corresponding to zero displacements. In this example, the third and fourth rows and columns correspond to zero displacements. Deleting these rows and columns, we obtain the global equations in the form $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$, where $[\mathbf{K}]$ is the global stiffness matrix:

$$\begin{bmatrix} 50 & -50 \\ -50 & 150 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} 40 \\ 0 \end{Bmatrix}.$$

The unknown displacements u_1 and u_2 can be obtained by solving the above equation as

$$u_1 = 1.2\,\text{cm} \text{ and } u_2 = 0.4\,\text{cm}.$$

By collecting all DOFs, the vector of nodal displacements can be obtained as: $\{\mathbf{Q}_s\}^{\mathrm{T}} = \{u_1, u_2, u_3, u_4\} = \{1.2,\ 0.4,\ 0,\ 0\}$.

Next, we substitute u_1 and u_2 into rows 3 and 4 in eq. (1.26) to calculate the reaction forces F_3 and F_4:

$$F_3 = 0u_1 - 30u_2 + 30u_3 + 0u_4 = -12 \text{ N},$$
$$F_4 = 0u_1 - 70u_2 + 0u_3 + 70u_4 = -28 \text{ N}.$$

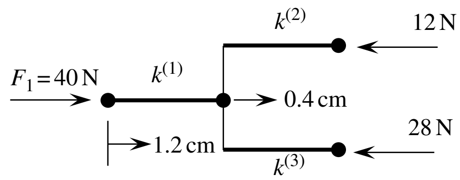


Figure 1.10 Free-body diagram of the structure

Based on the results obtained, we can now redraw the free-body diagram of the system, as shown in figure 1.10. Both reaction forces are in the negative x direction, and the sum of reactions is equal to the applied force in the opposite direction. ■

1.2.6 FE Program Organization

As the finite element analysis follows a standard procedure as described in the preceding section, it is possible to make a general-purpose FE program. Commercial FE programs typically consist of three parts: preprocessor, FE solver, and postprocessor. A preprocessor allows the user to define the structure, divide it into a number of elements, identify the nodes and their coordinates, define connectivity between various elements, and define material properties and the loads. Developments in computer graphics and CAD technology have resulted in sophisticated preprocessors that let the users create models and define various properties interactively on the computer terminal itself. A postprocessor takes the FE analysis results and presents them in a user-friendly graphical form. Again, developments in software and graphics have resulted in very sophisticated animations to help the analysts better understand the results of an FE model. This book is mostly concerned with the principles involved in the development and operation of the core FE program, which computes the stiffness matrix and assembles and solves the final set of equations. More on this is discussed in chapter 9. In addition, a brief introduction is provided to perform finite element analysis using commercial programs in the companion website of the book, where various finite element analysis programs are introduced, including Abaqus, ANSYS, Autodesk Nastran, and MATLAB Toolbox.

1.3 PLANE TRUSS ELEMENTS

This section presents the formulation of stiffness matrix and general procedures for solving the two-dimensional or plane truss using the direct stiffness method. A truss consisting of two elements is used to illustrate the solution procedures.

Consider the plane truss consisting of two bar elements or members as shown in figure 1.11. Two bars are connected with each other and with the ground using a pin joint; that is, their motion is constrained but free to rotate. A horizontal force $F = 50$ N is applied at the top node. Although the elements

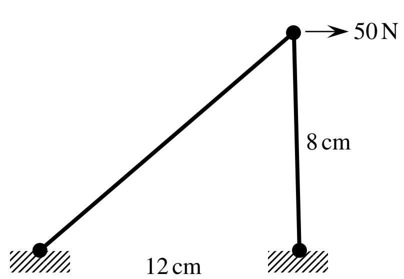


Figure 1.11 A plane truss consisting of two members

of the truss are uniaxial bars, the methods described in the previous section cannot be readily applied to this problem for two reasons: the two elements are not in the same direction but are inclined at different angles, and the external forces at a node can be applied in both x and y directions.

However, the element stiffness matrix of uniaxial bar elements will be applied to individual elements of the truss if we consider a local coordinate system. For a plane truss element, the following two coordinate systems can be defined:

1. The global coordinate system, x–y for the entire structure.
2. A local coordinate system, $\bar{x}-\bar{y}$ for a particular element such that the $\bar{x}$-axis is along the length of the element.

Referring to figure 1.12, the force-displacement relation of a truss element can be written in the local coordinate system as

$$\begin{Bmatrix} f_{1\bar{x}} \\ f_{2\bar{x}} \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{u}_1 \\ \bar{u}_2 \end{Bmatrix}, \tag{1.27}$$

where E, A, and L, respectively, are the Young's modulus, the area of the cross section, and the length of the element, and EA/L corresponds to the spring constant k in eq. (1.16).

Note that the forces and displacements are represented in the local coordinate system. In order to make the above equation more general, let us consider the transverse displacement $\bar{v}_1$ and $\bar{v}_2$ in the $\bar{y}$ direction. Corresponding transverse forces at each node can be defined as $f_{1\bar{y}}$ and $f_{2\bar{y}}$. However, in the truss element, these forces do not exist, and hence they are equated to zero. This is because the truss is a two-force member, where the member can only support a force in the axial direction. Then, the above stiffness matrix (system equations in matrix form) can be expanded to incorporate the forces and displacements in the $\bar{y}$ direction as shown below.

$$\begin{Bmatrix} f_{1\bar{x}} \\ f_{1\bar{y}} \\ f_{2\bar{x}} \\ f_{2\bar{y}} \end{Bmatrix} = \frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}\begin{Bmatrix} \bar{u}_1 \\ \bar{v}_1 \\ \bar{u}_2 \\ \bar{v}_2 \end{Bmatrix}. \tag{1.28}$$

The expanded local stiffness matrix in the above equation:

1. is a square matrix;
2. is symmetric; and
3. has diagonal elements that are greater than or equal to zero.

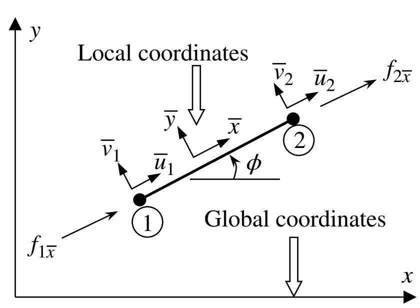


Figure 1.12 Local and global coordinate systems

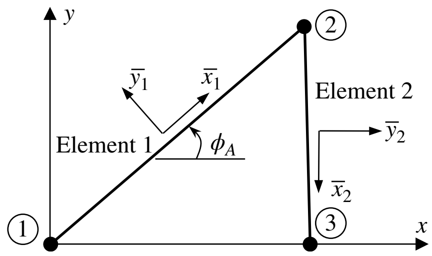


Figure 1.13 Local coordinate systems of the two-bar truss

The above stiffness matrix is valid only for the particular element 1 in the above example. It cannot be applied to other elements because the local coordinates $\bar{x}$–$\bar{y}$ are different for different elements. The local coordinates for element 2 are shown in figure 1.13.

In order to develop a system of equations that connect all elements in the truss, we need to transform the force-displacement relations, for instance, as shown in eq. (1.28), to the global coordinates, which is common for all elements of the truss. This requires the use of vector coordinate transformation.

1.3.1 Coordinate Transformation

As forces and displacements are vectors, we can use the vector transformation to find the relation between the displacements in local and global coordinates at a node. The global coordinate system, x–y, is fixed in space and common for the entire structure. On the other hand, the local coordinate system is parallel to the element. The local $\bar{x}$-axis is defined from node i to node j, while the local $\bar{y}$-axis is rotated by 90 degrees in the counterclockwise direction in the plane. Then, the angle ϕ for the element local coordinate is defined from the positive $\bar{x}$-axis to the positive x-axis.

Using the angle of the element, the relation between the displacements in local and global coordinates at node 1 can be written as

$$\boxed{\begin{Bmatrix} \bar{u}_1 \\ \bar{v}_1 \end{Bmatrix} = \begin{bmatrix} \cos\phi & \sin\phi \\ -\sin\phi & \cos\phi \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \end{Bmatrix}}.$$

A similar relation for node 2 will be

$$\boxed{\begin{Bmatrix} \bar{u}_2 \\ \bar{v}_2 \end{Bmatrix} = \begin{bmatrix} \cos\phi & \sin\phi \\ -\sin\phi & \cos\phi \end{bmatrix} \begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix}}.$$

Actually, we can combine the above relations for the two nodes to obtain the following relationship:

$$\underbrace{\begin{Bmatrix} \bar{u}_1 \\ \bar{v}_1 \\ \bar{u}_2 \\ \bar{v}_2 \end{Bmatrix}}_{\text{local}} = \begin{bmatrix} \cos\phi & \sin\phi & 0 & 0 \\ -\sin\phi & \cos\phi & 0 & 0 \\ 0 & 0 & \cos\phi & \sin\phi \\ 0 & 0 & -\sin\phi & \cos\phi \end{bmatrix} \underbrace{\begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix}}_{\text{global}}.$$

The above relation between local and global displacements can be written using a shorthand notation as

$$\{\bar{\mathbf{q}}\} = [\mathbf{T}]\{\mathbf{q}\}, \tag{1.29}$$

where $\{\bar{\mathbf{q}}\}$ and $\{\mathbf{q}\}$ are the element DOFs in the local and global coordinates, respectively, and $[\mathbf{T}]$ is the *transformation matrix*. In some literature $[\mathbf{T}]$ is called the rotation matrix. Since forces are also vectors, the forces $\{\bar{\mathbf{f}}\}$ in element coordinates are related to $\{\mathbf{f}\}$ in global coordinates as

$$\underbrace{\begin{Bmatrix} f_{1\bar{x}} \\ f_{1\bar{y}} \\ f_{2\bar{x}} \\ f_{2\bar{y}} \end{Bmatrix}}_{\text{local}} = \begin{bmatrix} \cos\phi & \sin\phi & 0 & 0 \\ -\sin\phi & \cos\phi & 0 & 0 \\ 0 & 0 & \cos\phi & \sin\phi \\ 0 & 0 & -\sin\phi & \cos\phi \end{bmatrix} \underbrace{\begin{Bmatrix} f_{1x} \\ f_{1y} \\ f_{2x} \\ f_{2y} \end{Bmatrix}}_{\text{global}},$$

or in shorthand notation

$$\{\bar{\mathbf{f}}\} = [\mathbf{T}]\{\mathbf{f}\}. \tag{1.30}$$

In the following section, we will express the local element equation (1.28) in the global coordinate using transformation relations in eqs. (1.29) and (1.30). Once all element equations are expressed in the global coordinate, they can be assembled using procedures similar to that of uniaxial bar elements.

1.3.2 Element Stiffness Matrix in the Global Coordinates

A key concept in finite element method is to discretize the entire system into many elements and to assemble them to make connections between elements. In order to make the assembly process valid, it is necessary that all DOFs in elements are represented in the same coordinate system. In this section, the element stiffness matrix in the local coordinates will be transformed into the global coordinates using the transformation relationship in the previous section.

For a single truss element, using the above coordinate transformation equation, we can proceed to transform the element stiffness matrix from local to global coordinates. Consider the truss element arbitrarily positioned in two-dimensional space as shown in figure 1.14. The element is defined in such a way that node 1 is the first node and node 2 is the second node. Therefore, the local $\bar{x}$-axis is defined from node 1 to node 2. The angle ϕ is defined from the positive x-axis to the positive $\bar{x}$-axis. If the element connectivity is defined from node 2 to node 1, then the angle should be defined as $\phi + 180°$. The stiffness of the element is given as $k = EA/L$.

The force-displacement equations can be expressed in the local coordinates as:

$$\begin{Bmatrix} f_{1\bar{x}} \\ f_{1\bar{y}} \\ f_{2\bar{x}} \\ f_{2\bar{y}} \end{Bmatrix} = \underbrace{\boxed{\frac{EA}{L}\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}}}_{\text{element stiffness matrix}} \begin{Bmatrix} \bar{u}_1 \\ \bar{v}_1 \\ \bar{u}_2 \\ \bar{v}_2 \end{Bmatrix}. \tag{1.31}$$

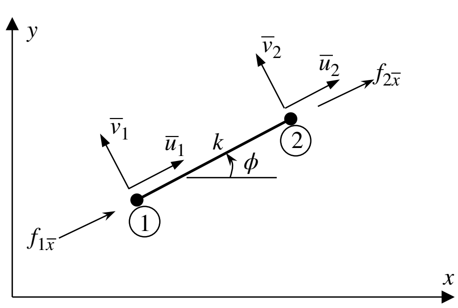


Figure 1.14 Definition of two-dimensional truss element

In the shorthand notation, eq. (1.31) takes the following form:

$$\{\bar{\mathbf{f}}\} = [\bar{\mathbf{k}}]\{\bar{\mathbf{q}}\}. \tag{1.32}$$

Substitution of eqs. (1.29) and (1.30) into eq. (1.31) yields

$$[\mathbf{T}]\{\mathbf{f}\} = [\bar{\mathbf{k}}][\mathbf{T}]\{\mathbf{q}\}.$$

Multiplying both sides of the equation by $[\mathbf{T}]^{-1}$,

$$\underbrace{\{\mathbf{f}\}}_{\text{global}} = [\mathbf{T}]^{-1}[\bar{\mathbf{k}}][\mathbf{T}]\underbrace{\{\mathbf{q}\}}_{\text{global}},$$

or

$$\{\mathbf{f}\} = [\mathbf{k}]\{\mathbf{q}\}. \tag{1.33}$$

The element stiffness matrix $[\mathbf{k}]$ in the global coordinates can now be expressed in terms of $[\bar{\mathbf{k}}]$ as

$$[\mathbf{k}] = [\mathbf{T}]^{-1}[\bar{\mathbf{k}}][\mathbf{T}]. \tag{1.34}$$

It can be shown that the inverse of the transformation matrix $[\mathbf{T}]$ is equal to its transpose, and hence $[\mathbf{k}]$ can be written as

$$[\mathbf{k}] = [\mathbf{T}]^{\mathrm{T}}[\bar{\mathbf{k}}][\mathbf{T}]. \tag{1.35}$$

Performing the matrix multiplication in eq. (1.35), we obtain an explicit expression for $[\mathbf{k}]$ as

$$[\mathbf{k}] = \frac{EA}{L}\begin{bmatrix} \cos^2\phi & \cos\phi\sin\phi & -\cos^2\phi & -\cos\phi\sin\phi \\ \cos\phi\sin\phi & \sin^2\phi & -\cos\phi\sin\phi & -\sin^2\phi \\ -\cos^2\phi & -\cos\theta\sin\phi & \cos^2\phi & \cos\phi\sin\phi \\ -\cos\phi\sin\phi & -\sin^2\phi & \cos\phi\sin\phi & \sin^2\phi \end{bmatrix}. \tag{1.36}$$

From eq. (1.36), it is clear that the element stiffness matrix of a plane truss element depends on the length L, axial rigidity EA, and the angle of orientation. As mentioned earlier, the element stiffness matrix is symmetric. Its determinant is equal to zero, and hence it does not have an inverse. Furthermore, the element stiffness matrix is positive semi-definite, and its diagonal elements are either equal to zero or greater than zero.

EXAMPLE 1.4 *Two-bar truss*

The two-bar truss shown in figure 1.15 has circular cross sections with a diameter of 0.25 cm and Young's modulus $E = 30 \times 10^6$ N/cm^2. An external force $F = 50$ N is applied in the horizontal direction at node 2. Calculate the displacement of each node and stress in each element.

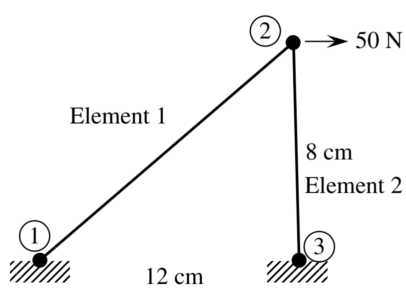


Figure 1.15 Two-bar truss structure

SOLUTION **Element 1:**

In the local coordinate system shown in figure 1.16, the force-displacement equations for element 1 is given in eq. (1.28), which can be transformed to the global coordinates similar to the one in eq. (1.33), to yield

$$\left\{\mathbf{f}^{(1)}\right\} = \left[\mathbf{k}^{(1)}\right]\left\{\mathbf{q}^{(1)}\right\}.$$

Since the orientation angle of the element is $\phi_1 = 33.7°$, the element equations in the global coordinates can be obtained using the stiffness matrix in eq. (1.36), as

$$\begin{Bmatrix} f_{1x}^{(1)} \\ f_{1y}^{(1)} \\ f_{2x}^{(1)} \\ f_{2y}^{(1)} \end{Bmatrix} = 102{,}150 \begin{bmatrix} 0.692 & 0.462 & -0.692 & -0.462 \\ 0.462 & 0.308 & -0.462 & -0.308 \\ -0.692 & -0.462 & 0.692 & 0.462 \\ -0.462 & -0.308 & 0.462 & 0.308 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{Bmatrix}.$$

Element 2:

For element 2, the same procedure can be applied with the orientation angle of the element being $\phi_2 = -90°$ (see figure 1.17). In the global coordinates, the element equations become

$$\begin{Bmatrix} f_{2x}^{(2)} \\ f_{2y}^{(2)} \\ f_{3x}^{(2)} \\ f_{3y}^{(2)} \end{Bmatrix} = 184{,}125 \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{Bmatrix} u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}.$$

Note that the orientation is measured in the counterclockwise direction from the positive x-axis of the global coordinates.

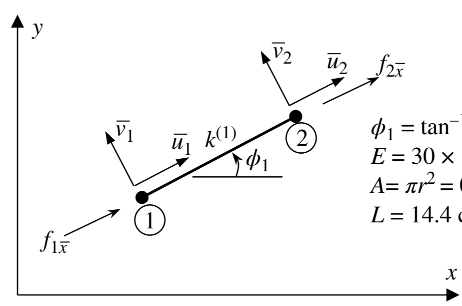


Figure 1.16 Local coordinates of element 1

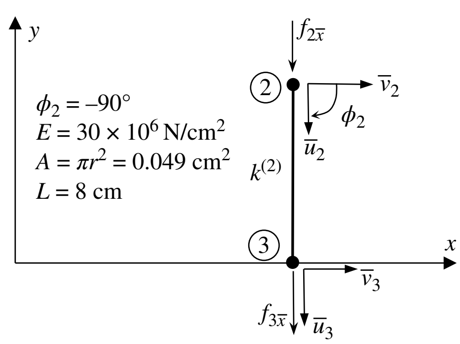


Figure 1.17 Local coordinates of element 2

Now we are ready to assemble the global stiffness matrix of the structure. Summing the two sets of force-displacement equations in the global coordinates:

Element 1

$$\begin{Bmatrix} F_{1x} \\ F_{1y} \\ F_{2x} \\ F_{2y} \\ F_{3x} \\ F_{3y} \end{Bmatrix} = \begin{bmatrix} 70687 & 47193 & -70687 & -47193 & 0 & 0 \\ 47193 & 31462 & -47193 & -31462 & 0 & 0 \\ -70687 & -47193 & 70687 & 47193 & 0 & \\ -47193 & -31462 & 47193 & 215587 & 0 & -184125 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -184125 & 0 & 184125 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \end{Bmatrix}.$$

Element 2

Note that two element stiffness matrices overlap at DOFs corresponding to u_2 and v_2 because the two elements are connected at node 2. Next, apply the following known boundary conditions:

(a) Nodes 1 and 3 are fixed; therefore, the displacement components of these two nodes are zero (u_1, v_1 and u_3, v_3).

(b) The only applied external forces are at node 2: $F_{2x} = 50$ N, and $F_{2y} = 0$ N.

$$\begin{Bmatrix} F_{1x} \\ F_{1y} \\ 50 \\ 0 \\ F_{3x} \\ F_{3y} \end{Bmatrix} = \begin{bmatrix} 70687 & 47193 & -70687 & -47193 & 0 & 0 \\ 47193 & 31462 & -47193 & -31462 & 0 & 0 \\ -70687 & -47193 & 70687 & 47193 & 0 & 0 \\ -47193 & -31462 & 47193 & 215587 & 0 & -184125 \\ 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -184125 & 0 & 184125 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ u_2 \\ v_2 \\ 0 \\ 0 \end{Bmatrix}.$$

We first delete the columns corresponding to zero displacements. In this example, the third and fourth columns correspond to nonzero displacements. We keep these two columns and strike out all other columns, to obtain

$$\begin{Bmatrix} F_{1x} \\ F_{1y} \\ 50 \\ 0 \\ F_{3x} \\ F_{3y} \end{Bmatrix} = \begin{bmatrix} -70687 & -47193 \\ -47193 & -31462 \\ 70687 & 47193 \\ 47193 & 215587 \\ 0 & 0 \\ 0 & -184125 \end{bmatrix} \begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix}. \tag{1.37}$$

F_{1x}, F_{1y}, F_{3x}, and F_{3y} are unknown reaction forces, and therefore we will delete the rows that correspond to these unknown reaction forces. We delete these rows because we want to keep only unknown displacement and known forces. We will use the deleted rows later to calculate unknown reaction forces after all displacements are calculated. Then, finally we have the following 2×2 matrix equation for the nodal displacements u_2 and v_2:

$$\begin{Bmatrix} 50 \\ 0 \end{Bmatrix} = \begin{bmatrix} 70687 & 47193 \\ 47193 & 215587 \end{bmatrix} \begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix}.$$

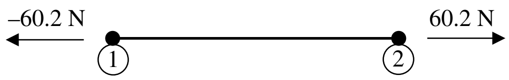


Figure 1.18 Element force for element 1 in local coordinates

Note that the global matrix equations only include unknown displacements and known forces. Since the global stiffness matrix in the above equation is positive definite, it is possible to invert it to solve for the unknown nodal displacements:

$$u_2 = 8.28 \times 10^{-4} \text{ cm},$$
$$v_2 = -1.81 \times 10^{-4} \text{ cm}.$$

Substituting the known u_2 and v_2 values into the matrix equation (1.37) and solve for the reaction forces:

$$\begin{Bmatrix} F_{1x} \\ F_{1y} \\ F_{3x} \\ F_{3y} \end{Bmatrix} = \begin{bmatrix} -70687 & -47193 \\ -47193 & -31462 \\ 0 & 0 \\ 0 & -184125 \end{bmatrix} \begin{Bmatrix} 8.28 \times 10^{-4} \\ -1.81 \times 10^{-4} \end{Bmatrix} = \begin{Bmatrix} -50 \\ -33.39 \\ 0 \\ 33.39 \end{Bmatrix} \text{N}.$$

Since the truss element is a two-force member, it is clear that the reaction force at node 3 is in the vertical direction, and the reaction force at node 1 is parallel to the direction of the element.

Once all displacements and forces are calculated in the global coordinates, it is necessary to go back to the element level in order to calculate element forces. However, it is important to note that the basic element behavior is expressed in the element local coordinates. Therefore, there are two steps involved in calculating element forces. First, among the vector of global displacements, $\{\mathbf{Q}_s\}$, it is necessary to extract displacements that belong to the element, $\{\mathbf{q}^{(e)}\}$. Then, it is necessary to transform the element displacements in the global coordinates into the local coordinates, $\{\bar{\mathbf{q}}^{(e)}\}$. For example, the nodal displacements of element 1 in the local coordinate system can be obtained from eq. (1.29), as

$$\begin{Bmatrix} \bar{u}_1 \\ \bar{v}_1 \\ \bar{u}_2 \\ \bar{v}_2 \end{Bmatrix} = \begin{bmatrix} .832 & .555 & 0 & 0 \\ -.555 & .832 & 0 & 0 \\ 0 & 0 & .832 & .555 \\ 0 & 0 & -.555 & .832 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 5.89 \times 10^{-4} \\ -6.11 \times 10^{-4} \end{Bmatrix}.$$

Then, the local force-displacement equations (1.32) can be used to calculate the element forces, as

$$\begin{Bmatrix} f_{1\bar{x}} \\ f_{1\bar{y}} \\ f_{2\bar{x}} \\ f_{2\bar{y}} \end{Bmatrix} = \frac{EA}{L} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} 0 \\ 0 \\ 5.89 \times 10^{-4} \\ -6.11 \times 10^{-4} \end{Bmatrix} = \begin{Bmatrix} -60.2 \\ 0 \\ 60.2 \\ 0 \end{Bmatrix} \text{N}. \tag{1.38}$$

Equation (1.38) represents the forces acting on the element in the local coordinate system. As expected, there is no force component in the $\bar{y}$ direction (local y direction). In the $\bar{x}$ direction (local x direction), the two nodes have the same magnitude of internal forces but in the opposite direction. As can be seen in figure 1.18, two equal and opposite forces act on the truss element, which results in tensile stresses in the element. In general, the sign of the force $f_{j\bar{x}}$ at node j (second node) will be the same as the sign of the element force, P. In the present example, the element force of element 1 is positive and has the magnitude of 60.2 N. Therefore, the normal stress in element 1 is tensile, and it can be calculated as $(60.2 / 0.049) = 1228 \text{ N/cm}^2$. ■

Another method to calculate the element force in a truss element is described below. This follows the method used in deriving eq. (1.22). For a truss element at an arbitrary orientation, eq. (1.22) is modified as

$$P^{(e)} = \left(\frac{AE}{L}\right)^{(e)} \Delta^{(e)} = \left(\frac{AE}{L}\right)^{(e)} \left(\bar{u}_j - \bar{u}_i\right). \tag{1.39}$$

Using the transformation relations in eq. (1.29), one can express the displacements in global coordinates. Let $l = \cos\phi$ and $m = \sin\phi$. Then, the expression for P takes the following form:

$$
\begin{aligned}
P^{(e)} &= \left(\frac{AE}{L}\right)^{(e)} \left[\left(lu_j + mv_j\right) - \left(lu + mv_i\right)\right] \\
&= \left(\frac{AE}{L}\right)^{(e)} \left[l\left(u_j - u_i\right) + m\left(v_j - v_i\right)\right].
\end{aligned}
\tag{1.40}
$$

In the above example, the following basic principles of FE analysis were used: (1) derive the force-displacement relations of each truss member, (2) assemble the equations to obtain the global equations, and (3) solve for unknown displacements. However, in practical problems with a large number of elements, one need not write the equations of equilibrium for each element. We would like to develop a systematic procedure that is suitable for a large number of elements. In this method, each element is assigned a first node and second node. These node numbers are denoted by i and j, respectively. The choice of the first and second nodes is arbitrary; however, it has to be consistent throughout the solution of the problem. The orientation of the element is defined by the angle the direction i–j makes with the positive x-axis, and it is denoted by ϕ. The direction cosines of the element are: $l = \cos\phi, m = \sin\phi$. We assign the row and column addresses to the element stiffness matrix as shown below. The element stiffness matrix in eq. (1.36) can be written in terms of l and m with the row and column addresses as

$$
[\mathbf{k}] = \left(\frac{EA}{L}\right)^{(e)}
\begin{array}{c}
\begin{matrix} u_i & v_i & u_j & v_j \end{matrix} \\
\begin{bmatrix}
l^2 & lm & -l^2 & -lm \\
lm & m^2 & -lm & -m^2 \\
-l^2 & -lm & l^2 & lm \\
-lm & -m^2 & lm & m^2
\end{bmatrix}
\end{array}
\begin{matrix} u_i \\ v_i \\ u_j \\ v_j \end{matrix}.
\tag{1.41}
$$

As illustrated in the following example, the row and column addresses are useful in assembling the element stiffness matrices into the global stiffness matrix. Note that the row addresses are the transpose of the column addresses.

EXAMPLE 1.5 *Plane truss with three elements*

The plane truss shown in figure 1.19 consists of three members connected to each other and to the walls by pin joints. The members make equal angles with each other, and element 2 is vertical. The members are identical to each other with the following properties: Young's modulus $E = 206 \times 10^9$ Pa, cross-sectional area $A = 1 \times 10^{-4}$ m^2, and length

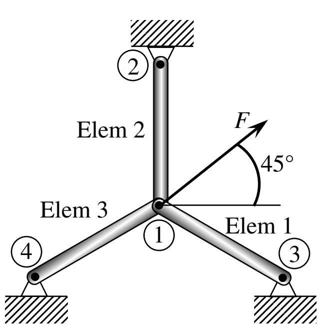


Figure 1.19 Plane structure with three truss elements

$L = 1$ m. An inclined force $F = 20{,}000$ N is applied at node 1. Solve for the displacements at Node 1 and stresses in the three elements.

SOLUTION Based on the above figure, a connectivity table that includes the element properties, node connectivity, and direction cosines can be calculated as shown in table 1.2.

Then, using eq. (1.36), the element stiffness matrices in the global coordinates system can be obtained as

$$\left[\mathbf{k}^{(1)}\right] = 206\times10^5 \begin{bmatrix} 0.750 & -0.433 & -0.750 & 0.433 \\ -0.433 & 0.250 & 0.433 & -0.250 \\ -0.750 & 0.433 & 0.750 & -0.433 \\ 0.433 & -0.250 & -0.433 & 0.250 \end{bmatrix} \begin{matrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{matrix},$$

$$\left[\mathbf{k}^{(2)}\right] = 206\times10^5 \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{matrix} u_1 \\ v_1 \\ u_2 \\ v_2 \end{matrix},$$

$$\left[\mathbf{k}^{(3)}\right] = 206\times10^5 \begin{bmatrix} 0.750 & 0.433 & -0.750 & -0.433 \\ 0.433 & 0.250 & -0.433 & -0.250 \\ -0.750 & -0.433 & 0.750 & 0.433 \\ -0.433 & -0.250 & 0.433 & 0.250 \end{bmatrix} \begin{matrix} u_4 \\ v_1 \\ u_4 \\ v_4 \end{matrix}.$$

Note that row addresses are indicated on the RHS of the stiffness matrices, and they are useful in assembling the structural stiffness matrix. One can easily identify the column addresses, although they are not written above the stiffness matrices. The structural stiffness matrix and the finite element equations are obtained by assembling the three element stiffness matrices:

$$206\times10^5 \begin{bmatrix} 1.5 & 0 & 0 & 0 & -0.750 & 0.433 & -0.750 & -0.433 \\ & 1.5 & 0 & -1 & 0.433 & -0.250 & -0.433 & -0.250 \\ & & 0 & 0 & 0 & 0 & 0 & 0 \\ & & & 1 & 0 & 0 & 0 & 0 \\ & & & & 0.750 & -0.433 & 0 & 0 \\ & \text{Symmetric} & & & & 0.250 & 0 & 0 \\ & & & & & & 0.750 & 0.433 \\ & & & & & & & 0.250 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_3 \\ v_3 \\ u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} F_{x1} \\ F_{y1} \\ F_{x2} \\ F_{y2} \\ F_{x3} \\ F_{y3} \\ F_{x4} \\ F_{y4} \end{Bmatrix},$$

or

$$[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}.$$

Table 1.2 Connectivity table with element properties for example 1.5

Element	AE/L	LN1 (i)	LN2 (j)	ϕ	$l = \cos\phi$	$m = \sin\phi$
1	206×10^5	1	3	$-\pi/6$	0.866	−0.5
2	206×10^5	1	2	$\pi/2$	0	1
3	206×10^5	1	4	$-5\pi/6$	−0.866	−0.5

Now, the known external forces and displacements are applied to the above matrix equation. First, the inclined force at node 1 is decomposed into x and y directions, as

$$F_{1x} = 20000 \cdot \cos(\pi/4) = 14{,}142$$
$$F_{1y} = 20000 \cdot \sin(\pi/4) = 14{,}142.$$

In addition, since nodes 2, 3, and 4 are fixed, their displacements are equal to zero:

$$u_2 = v_2 = u_3 = v_3 = u_4 = v_4 = 0.$$

We delete the rows and columns in matrix $[\mathbf{K}_s]$ corresponding to those DOFs that have zero displacements. Then, the global FE equations are written in the form $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$:

$$206 \times 10^5 \begin{bmatrix} 1.5 & 0 \\ 0 & 1.5 \end{bmatrix} \begin{Bmatrix} u_1 \\ v_1 \end{Bmatrix} = \begin{Bmatrix} 14142 \\ 14142 \end{Bmatrix}.$$

The global stiffness matrix is now positive-definite, and thus invertible. After solving for unknown displacements, we have

$$u_1 = 0.458\,\text{mm},$$
$$v_1 = 0.458\,\text{mm}.$$

By including all other zero displacements, the vector of nodal displacements can be written as $\{\mathbf{Q}_s\}^{\mathrm{T}} = \{0.458,\ 0.458,\ 0,\ 0,\ 0,\ 0,\ 0,\ 0\}$.

Force in each element can be obtained using eq. (1.40) and element properties given in table 1.2. For example, the force in element 1 is

$$P^{(1)} = 206 \times 10^5 (0.866(u - u_1) - 0.5(v - v_1)) = -3{,}450 \text{ N}.$$

The same calculation can be repeated for other elements to obtain:

$$P^{(2)} = -9{,}440 \text{ N},$$
$$P^{(3)} = 12{,}900 \text{ N}.$$

The negative values of $P^{(1)}$ and $P^{(2)}$ indicate compressive forces in those elements. The stresses of the elements can be obtained by dividing the force by the area of cross section:

$$\sigma^{(1)} = -34.5 \text{ MPa},$$
$$\sigma^{(2)} = -94.4 \text{ MPa},$$
$$\sigma^{(3)} = 129 \text{ MPa}.$$

Once element properties, node connectivity, and direction cosines of all elements are listed as in table 1.2, it is easy to make a computer program that can build the global stiffness matrix and solve for unknown displacements. ■

1.3.3 Method of Superposition

In deriving the truss equations, we have used two major assumptions. The displacements are small such that the elongation of an element or a member ΔL is much less than the original length L, that is, $\Delta L << L$, or the strain $\varepsilon = \Delta L/L << 1$. Another assumption is that the stress-strain relation is linear, that is, the Young's modulus is a constant. The above assumptions lead to a simple but useful principle called the superposition principle.

Consider a truss subjected to forces $\{\mathbf{F}\}$ at the nodes or joints. We assume that the truss is supported such that the rigid body displacements are completely constrained. The resulting displacements are $\{\mathbf{Q}\}$. The relation between the applied forces and displacements is given by

$$[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}, \tag{1.42}$$

where $\{\mathbf{K}\}$ is the global stiffness matrix. Now if all the forces are multiplied by a nonzero factor α such that the applied forces are $\{\alpha\mathbf{F}\}$, then the corresponding displacements can be derived as $\{\alpha\mathbf{Q}\}$, that is,

$$[\mathbf{K}]\{\alpha\mathbf{Q}\} = \{\alpha\mathbf{F}\}. \tag{1.43}$$

That is, the relationship between displacements and applied forces is linear. If we double the forces, then the displacements will also be doubled. The above result is obvious as the coefficients of stiffness matrix $\{\mathbf{K}\}$ are constants and depend only the truss geometry and Young's modulus of the material. Note that the above factor α can be negative.

In addition to the linearity between displacements and applied forces, the relationship between displacement and strain is linear, as well as the relationship betweeh strain and stress. Therefore, if the force is doubled, then the displacements, strains, and stresses will also be doubled. This can be a very useful tool for design. For example, let us assume that the maximum stress under the current load is 500 MPa, and we want to limit the maximum stress to 300 MPa. Then it is straightforward that we need to reduce the applied force by 40%.

The second principle that follows the above can be described as follows. Let $\{\mathbf{Q}^{(1)}\}$ and $\{\mathbf{Q}^{(2)}\}$ be the displacements for two different sets of loads $\{\mathbf{F}^{(1)}\}$ and $\{\mathbf{F}^{(2)}\}$, respectively. If the loads are applied together with different factors, that is, if the load is given by $\left\{\alpha\mathbf{F}^{(1)} + \beta\mathbf{F}^{(2)}\right\}$, then the resulting displacements will be the sum of the two sets of displacements, $\left\{\alpha\mathbf{Q}^{(1)} + \beta\mathbf{Q}^{(2)}\right\}$. Again the above result is obvious, as

$$[\mathbf{K}]\left\{\alpha\mathbf{Q}^{(1)} + \beta\mathbf{Q}^{(2)}\right\} = \left\{\alpha\mathbf{F}^{(1)} + \beta\mathbf{F}^{(2)}\right\}. \tag{1.44}$$

In the view of eq. (1.44), it is enough to apply a unit force and calculating resulting stresses, which is called stress influence coefficient. When the actual magnitude of the force is given, the actual stress can be calculated by multiplying the stress influence coefficients by the magnitude of the force.

EXAMPLE 1.6 *Superposition of multiple loads*

Consider the plane truss in example 1.5. Let $\{\mathbf{F}^{(1)}\}$ be a unit force in the x direction at node 1 and $\{\mathbf{F}^{(2)}\}$ be a unit force in the y direction. Show that the displacements using the superposition principle, $\{\mathbf{Q}\} = \alpha\left\{\mathbf{Q}^{(1)}\right\} + \beta\left\{\mathbf{Q}^{(2)}\right\}$, is the same as the actual displacements in Example 1.5, where $\alpha = \beta = 10{,}000\sqrt{2}$.

SOLUTION The element stiffness matrices and the assembled global stiffness matrix can be found in Example 1.5. After applying the boundary conditions, the global matrix equations for $\{\mathbf{F}^{(1)}\}$ can be written as

$$206 \times 10^5 \begin{bmatrix} 1.5 & 0 \\ 0 & 1.5 \end{bmatrix} \begin{Bmatrix} u_1^{(1)} \\ v_1^{(1)} \end{Bmatrix} = \begin{Bmatrix} 1 \\ 0 \end{Bmatrix}.$$

After solving for unknown displacements, we have

$$\left\{\mathbf{Q}^{(1)}\right\} = \begin{Bmatrix} u_1^{(1)} \\ v_1^{(1)} \end{Bmatrix} = \begin{Bmatrix} 0.3236 \\ 0.0 \end{Bmatrix} \times 10^{-7}.$$

Also, when the vertical force $\{\mathbf{F}^{(2)}\}$ is applied,

$$206\times10^5\begin{bmatrix}1.5 & 0\\ 0 & 1.5\end{bmatrix}\begin{Bmatrix}u_1^{(2)}\\ v_1^{(2)}\end{Bmatrix}=\begin{Bmatrix}0\\ 1\end{Bmatrix}.$$

After solving for unknown displacements, we have

$$\left\{\mathbf{Q}^{(2)}\right\}=\begin{Bmatrix}u_1^{(2)}\\ v_1^{(2)}\end{Bmatrix}=\begin{Bmatrix}0.0\\ 0.3236\end{Bmatrix}\times10^{-7}.$$

Therefore, the displacements caused by the combined force with $\alpha=\beta=10{,}000\sqrt{2}$ can be obtained as

$$\{\mathbf{Q}\}=\alpha\left\{\mathbf{Q}^{(1)}\right\}+\beta\left\{\mathbf{Q}^{(2)}\right\}=\begin{Bmatrix}0.458\\ 0.458\end{Bmatrix}\text{mm},$$

which is the same result than that of example 1.5. It is a good practice to show that the same superposition can be applied to the stress. ■

1.4 THREE-DIMENSIONAL TRUSS ELEMENTS (SPACE TRUSS)

This section presents the formulation of the stiffness matrix and general procedures for solving the three-dimensional or space truss using the direct stiffness method. The procedure is similar to that of plane truss except that an additional DOF is added at each node.

1.4.1 Three-Dimensional Coordinate Transformation

The coordinate transformation used for the two-dimensional truss element can be generalized to the three-dimensional truss elements for space trusses.

A space-truss element has three DOFs, u, v, and w, at each node. Thus, the space–truss element is a 2-node 6-DOF element. Corresponding to the three displacements at each node, there are three forces, f_x, f_y, and f_z. Similar to the two-dimensional plane truss, space truss elements are connected with other elements or the ground using a ball-and-socket joint; that is, two elements connected at a node can rotate freely with respect to each other. The displacements and forces can also be expressed in a local or elemental coordinate system $\bar{x}$–$\bar{y}$–$\bar{z}$ as shown in figure 1.20. The forces and displacements in the local coordinate system are related by the same equation as for two dimensions and is therefore similar to eq. (1.27).

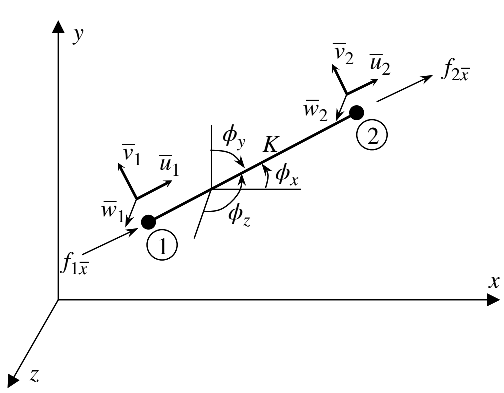


Figure 1.20 Three-dimensional coordinates transformation

$$\begin{Bmatrix} f_{i\bar{x}} \\ f_{j\bar{x}} \end{Bmatrix} = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{u}_i \\ \bar{u}_j \end{Bmatrix},$$

or

$$\{\bar{\mathbf{f}}\} = [\bar{\mathbf{k}}]\{\bar{\mathbf{q}}\}. \tag{1.45}$$

In order to assemble truss elements, the element equation (1.45) must be transformed into the global coordinate system. For the plane truss element in eq. (1.31), we expanded the force-displacement relation to a 4×4 matrix equation by including two transverse displacements. In the same way, it is possible to expand the force-displacement relation to a 6×6 matrix equation for space truss, and the transformation matrix would also be a 6×6 matrix, which is a big matrix to handle. Instead, the same result can be obtained if we keep 2×2 matrix as in eq. (1.45) and the following 2×6 transformation matrix:

$$\begin{Bmatrix} \bar{u}_i \\ \bar{u}_j \end{Bmatrix} = \begin{bmatrix} l & m & n & 0 & 0 & 0 \\ 0 & 0 & 0 & l & m & n \end{bmatrix}\begin{Bmatrix} u_i \\ v_i \\ w_i \\ u_j \\ v_j \\ w_j \end{Bmatrix},$$

or

$$\{\bar{\mathbf{q}}\} = [\mathbf{T}]\{\mathbf{q}\}, \tag{1.46}$$

where l, m, and n are the direction cosines of the element connecting nodes i and j, which can be calculated from the element length L and the nodal coordinates as shown below:

$$\begin{aligned} &l = \frac{x_j - x_i}{L}, \quad m = \frac{y_j - y_i}{L}, \quad n = \frac{z_j - z_i}{L}, \\ &L = \sqrt{(x_j - x_i)^2 + (y_j - y_i)^2 + (z_j - z_i)^2}. \end{aligned} \tag{1.47}$$

The direction cosines in eq. (1.47) are basically the same as the direction cosines in the plane truss element, where we used the definition of $l = \cos\phi$ and $m = \sin\phi$. But, the actual definitions of direction cosines were $l = \cos\phi_x$ and $m = \cos\phi_y$ where ϕ_x is the angle between the positive local $\bar{x}$-axis to the positive global x-axis, and ϕ_y is the angle between the positive local $\bar{x}$-axis to the positive global y-axis. To compare with the plane truss case in figure 1.14, we can use the property that $\phi_x = \phi$ and $\phi_y = 90^\circ - \phi$.

Similar to transformation for displacements, the nodal forces in the local coordinate system can be transformed as

$$\begin{Bmatrix} f_{ix} \\ f_{iy} \\ f_{iz} \\ f_{jx} \\ f_{jy} \\ f_{jz} \end{Bmatrix} = \begin{bmatrix} l & 0 \\ m & 0 \\ n & 0 \\ 0 & l \\ 0 & m \\ 0 & n \end{bmatrix}\begin{Bmatrix} f_{i\bar{x}} \\ f_{j\bar{x}} \end{Bmatrix},$$

or

$$\{\mathbf{f}\} = [\mathbf{T}]^{\mathrm{T}}\{\bar{\mathbf{f}}\}. \tag{1.48}$$

1.4.2 Element Stiffness Matrix in the Global Coordinates

Substituting for $\{\bar{\mathbf{q}}\}$ from eq. (1.46) into eq. (1.45) and post-multiplying both sides of the equation by $[\mathbf{T}]^{\mathrm{T}}$, we obtain

$$[\mathbf{T}]^{\mathrm{T}}\{\bar{\mathbf{f}}\} = [\mathbf{T}]^{\mathrm{T}}[\bar{\mathbf{k}}][\mathbf{T}]\{\mathbf{q}\},$$

or

$$\{\mathbf{f}\} = [\mathbf{k}]\{\mathbf{q}\}, \tag{1.49}$$

where $[\mathbf{k}]$ is the element stiffness matrix that relates the nodal forces and displacements expressed in the global coordinates. From eq. (1.49) it is clear that $[\mathbf{k}]$ is obtained as the product $[\mathbf{T}]^{\mathrm{T}}[\bar{\mathbf{k}}][\mathbf{T}]$. An explicit form of $[\mathbf{k}]$ is given below.

$$[\mathbf{k}] = \frac{EA}{L}\begin{bmatrix} l^2 & lm & ln & -l^2 & -lm & -ln \\ & m^2 & mn & -lm & -m^2 & -mn \\ & & n^2 & -ln & -mn & -n^2 \\ & & & l^2 & lm & ln \\ & \text{sym} & & & m^2 & mn \\ & & & & & n^2 \end{bmatrix} \begin{matrix} u_i \\ v_i \\ w_i \\ u_j \\ v_j \\ w_j \end{matrix}, \tag{1.50}$$

where the row DOFs are shown next to the matrix. Assembling $[\mathbf{k}]$ into the structural stiffness matrix $[\mathbf{K_s}]$ and then deleting the rows and columns to obtain the global stiffness matrix $[\mathbf{K}]$ follow procedures similar to those of plane truss.

EXAMPLE 1.7 ***Space truss***

Use the FEM to determine the displacements and forces in the space truss, shown in figure 1.21. The coordinates of the nodes in meter units are given in table 1.3. Assume Young's modulus $E = 70$ GPa and area of cross section $A = 1\ \text{cm}^2$. The magnitude of the downward force (negative z direction) at node 4 is equal to 10,000 N.

SOLUTION The first step is to determine the direction cosines of the elements. Their length and direction cosines are calculated using the formulas in eq. (1.47). Table 1.4 shows the connectivity and the direction cosines of all the elements.

Using eq. (1.50), element stiffness matrices can be constructed. Since all nodes are fixed except for node 4, the rows and columns corresponding to zero DOF can be deleted at this stage, and the element stiffness matrix contains only non-fixed DOFs. In the case of element 1, for example, the 6×6 element stiffness matrix in eq. (1.50) involves

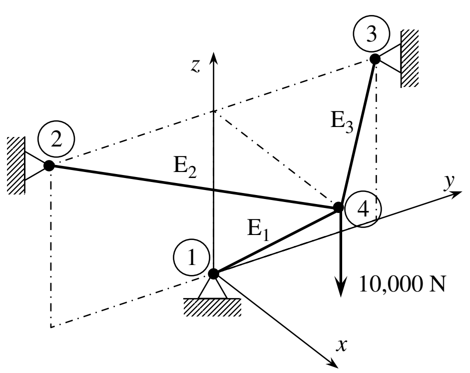


Figure 1.21 Three-bar space truss structure

Table 1.3 Nodal coordinates of space truss structure in example 1.6

Node	x	y	z
1	0	0	0
2	0	–1	1
3	0	1	1
4	1	0	1

Table 1.4 Element connectivity and direction cosines for truss structure in figure 1.21

Element	LN1 (i)	LN2 (j)	L (m)	l	m	n
1	1	4	$\sqrt{2}$	$\sqrt{2}/2$	0	$\sqrt{2}/2$
2	2	4	$\sqrt{2}$	$\sqrt{2}/2$	$\sqrt{2}/2$	0
3	3	4	$\sqrt{2}$	$\sqrt{2}/2$	$-\sqrt{2}/2$	0

two nodes, $i = 1$ and $j = 4$. Since node 1 is fixed, the three rows and columns that correspond to Node 1 can be deleted at the element level. Then, the 3×3 reduced element stiffness matrix can be obtained. By repeating the procedure for all three elements, we can obtain the following reduced element stiffness matrices:

$$\left[\mathbf{k}^{(1)}\right] = 35\sqrt{2} \times 10^5 \begin{bmatrix} 0.5 & 0 & 0.5 \\ 0 & 0 & 0 \\ 0.5 & 0 & 0.5 \end{bmatrix} \begin{matrix} u_4 \\ v_4 \\ w_4 \end{matrix},$$

$$\left[\mathbf{k}^{(2)}\right] = 35\sqrt{2} \times 10^5 \begin{bmatrix} 0.5 & 0.5 & 0 \\ 0.5 & 0.5 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_4 \\ v_4 \\ w_4 \end{matrix},$$

$$\left[\mathbf{k}^{(3)}\right] = 35\sqrt{2} \times 10^5 \begin{bmatrix} 0.5 & -0.5 & 0 \\ -0.5 & 0.5 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_4 \\ v_4 \\ w_4 \end{matrix}.$$

After assembly, we obtain the global equations in the form $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$:

$$35\sqrt{2} \times 10^5 \begin{bmatrix} 1.5 & 0 & 0.5 \\ 0 & 1.0 & 0 \\ 0.5 & 0 & 0.5 \end{bmatrix} \begin{Bmatrix} u_4 \\ v_4 \\ w_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -10{,}000 \end{Bmatrix}.$$

The above global stiffness matrix is positive definite as the displacement boundary conditions have already been implemented. By solving the global matrix equation, the unknown nodal displacements are obtained as

$$u_4 = 2.020 \times 10^{-3}\,\text{m},$$
$$v_4 = 0,$$
$$w_4 = -6.061 \times 10^{-3}\,\text{m}.$$

In order to calculate the element forces, the displacements of nodes of each element have to be transformed to the local coordinates using the relation $\{\bar{\mathbf{q}}\} = [\mathbf{T}]\{\mathbf{q}\}$ in eq. (1.46). Then, the element forces are calculated using

$\{\bar{\mathbf{f}}\} = [\bar{\mathbf{k}}]\{\bar{\mathbf{q}}\}$. The element force P can be obtained from the element force $f_{j\bar{x}}$ (the force in the second node) for the corresponding element. For element 1, the nodal displacements in the local coordinate system can be obtained as

$$\begin{Bmatrix} \bar{u}_1 \\ \bar{u}_4 \end{Bmatrix}^{(1)} = \frac{\sqrt{2}}{2}\begin{bmatrix} 1 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 1 \end{bmatrix}\begin{Bmatrix} u_1 = 0 \\ v_1 = 0 \\ w_1 = 0 \\ u_4 \\ v_4 \\ w_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -2.857 \end{Bmatrix} \times 10^{-3}\text{m}.$$

From the force-displacement relation, the element force can be obtained as

$$\begin{Bmatrix} f_{1\bar{x}} \\ f_{4\bar{x}} \end{Bmatrix}^{(1)} = \left(\frac{AE}{L}\right)^{(1)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{u}_1 \\ \bar{u}_4 \end{Bmatrix} = \begin{Bmatrix} 14{,}141 \\ -14{,}141 \end{Bmatrix}.$$

Thus, the element force is

$$P^{(1)} = (f_{4\bar{x}})^{(1)} = -14{,}141\,\text{N}.$$

The above calculations are repeated for elements 2 and 3:

$$\begin{Bmatrix} \bar{u}_2 \\ \bar{u}_4 \end{Bmatrix}^{(2)} = \frac{\sqrt{2}}{2}\begin{bmatrix} 1 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 1 & 0 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \\ w_2 \\ u_4 \\ v_4 \\ w_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 1.428 \end{Bmatrix} \times 10^{-3}\text{m}$$

$$\begin{Bmatrix} f_{2\bar{x}} \\ f_{4\bar{x}} \end{Bmatrix}^{(2)} = \left(\frac{AE}{L}\right)^{(2)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{u}_2 \\ \bar{u}_4 \end{Bmatrix} = \begin{Bmatrix} -7{,}070 \\ +7{,}070 \end{Bmatrix}$$

$$P^{(2)} = (f_{4\bar{x}})^{(2)} = +7{,}070\text{N}$$

$$\begin{Bmatrix} \bar{u}_3 \\ \bar{u}_4 \end{Bmatrix}^{(3)} = \frac{\sqrt{2}}{2}\begin{bmatrix} 1 & -1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & -1 & 0 \end{bmatrix}\begin{Bmatrix} u_3 \\ v_3 \\ w_3 \\ u_4 \\ v_4 \\ w_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 1.428 \end{Bmatrix} 10^{-3}\text{m}$$

$$\begin{Bmatrix} f_{3\bar{x}} \\ f_{4\bar{x}} \end{Bmatrix}^{(3)} = \left(\frac{AE}{L}\right)^{(3)}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} \bar{u}_2 \\ \bar{u}_4 \end{Bmatrix} = \begin{Bmatrix} -7{,}070 \\ +7{,}070 \end{Bmatrix}$$

$$P^{(3)} = (f_{4\bar{x}})^{(3)} = +7{,}070\text{N}.$$

Alternately, we can calculate the axial forces in an element using an equation similar to eq. (1.40). For a three-dimensional element, this equation takes the form:

$$P^{(e)} = \left(\frac{AE}{L}\right)^{(e)}\left(l\left(u_j - u_i\right) + m\left(v_j - v_i\right) + n\left(w_j - w_i\right)\right). \tag{1.51}$$

Note that element 1 is in compression, while elements 2 and 3 are in tension. ■

1.5 THERMAL STRESSES

Thermal stresses in structural elements appear when they are subjected to a temperature change from the reference temperature. At the reference temperature, as shown in figure 1.22(a), if there are no external loads acting on the structure, then there will be no stresses; stresses and strains vanish simultaneously. When the temperature of one or more elements in a structure is changed as shown in figure 1.22(b), then the members tend to expand. However, if the expansion is partially constrained by the surrounding members, then stresses will develop due to the constraint. The constraining members will also experience a force as a reaction to this constraint. This reaction, in turn, produces thermal stresses in the members. The same idea can be extended to thermal stresses in a solid if we imagine the solid to contain many small elements, and each restraining others from expanding or contracting due to temperature change.

The linear relation between stresses and strains for linear elastic solids is valid only when the temperature remains constant and in the absence of residual stresses. In the presence of a temperature differential, that is, when the temperature is different from the reference temperature, we need to use the thermo-elastic stress-strain relations. Such a relation in one dimension is:

$$\sigma = E(\varepsilon - \alpha \Delta T), \tag{1.52}$$

where σ is the uniaxial stress, ε is the total strain, E is Young's modulus, α is the coefficient of thermal expansion (CTE) and ΔT is the difference between the operating temperature and the reference temperature. From eq. (1.52) it is clear that the *reference temperature* is defined as the temperature at which both stress and strain vanish simultaneously when there is no external load. Equation (1.52) states that the stress is caused by and proportional to the mechanical strain, which is the difference between the total strain $\varepsilon = \Delta L/L$ and the *thermal strain* $\alpha\Delta T$. The strain-stress relation now takes the following form:

$$\varepsilon = \frac{\sigma}{E} + \alpha \Delta T. \tag{1.53}$$

The total strain is the sum of the mechanical strain caused by the stresses and the thermal strain caused by the temperature rise. Similarly, thermo-elastic stress-strain relations can be developed in two and three dimensions[4]. It is noted that only the mechanical strain can produce stress, but not the thermal strain. However, the finite element method can only solve for the total strain, not the mechanical strain. Therefore, in order to calculate stress correctly, it is necessary to subtract the thermal strain from the total strain.

The above stress-strain relation can be converted into the force-displacement relation by multiplying eq. (1.52) by the area of cross section of the uniaxial bar, A:

$$P = AE\left(\frac{\Delta L}{L} - \alpha \Delta T\right) = AE\frac{\Delta L}{L} - AE\alpha\Delta T, \tag{1.54}$$

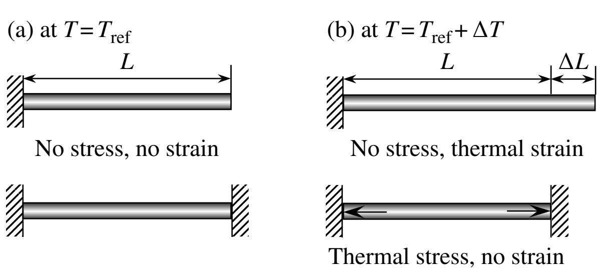


Figure 1.22 Effects of temperature change on the structure

---

[4] A.P. Boresi and R.J. Schmidt. *Advanced Mechanics of Materials*, Sixth Edition. John Wiley & Sons, Inc., New York, NY.

where the first term in the parentheses is the total strain or simply strain and the second, the thermal strain.

Before we introduce the formal method of solving a thermal stress problem using FE, it will be instructive to discuss the method of superposition for solving thermal stress problems.

1.5.1 Method of Superposition

Consider the truss shown in figure 1.23(a). Assume that the temperature of element 2 is raised by ΔT, and elements 1 and 3 remain at the reference temperature. There are no external forces acting at node 4. The objective is to compute the nodal displacements and forces that will be developed in each member. First, if all three elements are disconnected, then only element 2 will expand due to temperature change. Imagine that we apply a pair of equal and opposite forces on the two nodes of element 2, such that the forces restrain the thermal expansion of the element (see figure 1.23(b)). This force can be determined by setting $\Delta L = 0$ in eq. (1.54) and it is equal to $-AE\alpha\Delta T$. That is, a compressive force is required to prevent element 2 from expanding. Hence the force resultant on element 2, $P^{(2)}$, is compressive with magnitude equal to $AE\alpha\Delta T$. If several members are subjected to temperature changes, then a corresponding pair of forces is applied to each element.

The solution to this problem, which will be called problem I, is obvious: the nodal displacements are all equal to zero because no element is allowed to expand or contract, and the force in element 2 is equal to $-AE\alpha\Delta T$. There are no forces in elements 1 and 3. However, this is not the problem we want to solve. The pair of forces applied to element 2 was not there in the original problem. Hence, we have to remove these extraneous forces.

To remove the force shown in figure 1.23(b), we superpose the results from a problem II, where there is no thermal effect, but the forces applied in problem I are all reversed. This is depicted in figure 1.23(c). Sometimes the forces acting in problem II are called fictitious thermal forces, as they do not actually exist. Problem II is a standard truss problem, and hence the FEM we have already discussed can be used to determine the nodal displacements and the element forces. The solution of the problem in figure 1.23(a) can be obtained by adding the solutions from both problems I and II.

EXAMPLE 1.8 ***Thermal stresses in a plane truss***

Solve the nodal displacements and element forces of the plane truss problem in figure 1.23. Use the following numerical values: $AE = 10^7$ N, $L = 1$ m, $\alpha = 10^{-5}/°C$, $\Delta T = 100$ °C.

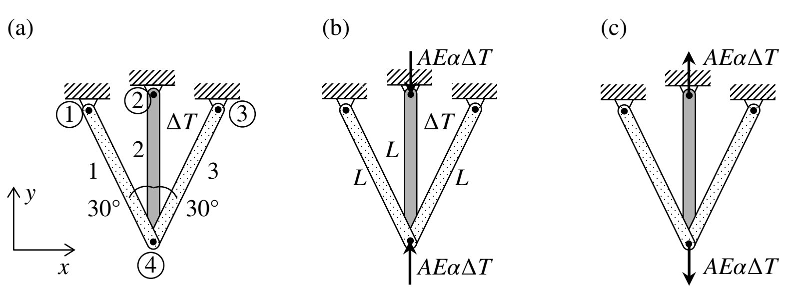


Figure 1.23 A three-element truss: (a) The middle element is subjected to a temperature rise. This is the given problem. (b) A pair of compressive forces is applied to element 2 to prevent it from expanding. This is called problem I. (c) The forces in problem I are reversed. No thermal stresses are involved in this problem. This is called problem II.

Table 1.5 Element connectivity and direction cosines for truss structure in figure 1.23

Element	LN1 (i)	LN2 (j)	AE/L (N/m)	$AE\alpha\Delta T$ (N)	ϕ (degrees)	$l=\cos\phi$	$m=\sin\phi$
1	1	4	10^7	0	–60	1/2	$-\sqrt{3}/2$
2	2	4	10^7	10,000	–90	0	–1
3	3	4	10^7	0	240	–1/2	$-\sqrt{3}/2$

SOLUTION The solution to problem I is as follows:

$$u_4 = v_4 = 0,$$
$$P^{(1)} = 0,\ P^{(2)} = -AE\alpha\Delta T = -10{,}000\,\text{N},\ P^{(3)} = 0.$$

Problem II is depicted in figure 1.23(c). The element properties and direction cosines are listed in table 1.5.

The element stiffness matrices in the global coordinates are written below. For convenience, the rows and columns corresponding to zero DOFs are deleted, and only those corresponding to active DOFs are shown.

$$[\mathbf{k}^{(1)}] = \frac{10^7}{4}\begin{bmatrix} 1 & -\sqrt{3} \\ -\sqrt{3} & 3 \end{bmatrix}\begin{matrix} u_4 \\ v_4 \end{matrix},$$
$$[\mathbf{k}^{(2)}] = 10^7\begin{bmatrix} 0 & 0 \\ 0 & 1 \end{bmatrix}\begin{matrix} u_4 \\ v_4 \end{matrix},$$
$$[\mathbf{k}^{(3)}] = \frac{10^7}{4}\begin{bmatrix} 1 & \sqrt{3} \\ \sqrt{3} & 3 \end{bmatrix}\begin{matrix} u_4 \\ v_4 \end{matrix}.$$

Assembling the element stiffness matrices, we obtain the global stiffness matrix [**K**]. The only external force for this problem is $F_{4y} = -10{,}000$ N.

$$\frac{10^7}{4}\begin{bmatrix} 2 & 0 \\ 0 & 10 \end{bmatrix}\begin{Bmatrix} u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10{,}000 \end{Bmatrix}.$$

Since the global stiffness matrix is positive definite, the solution for displacements can be obtained as:

$$u_4 = 0,$$
$$v_4 = -0.4\times 10^{-3}\,\text{m}.$$

Table 1.6 Solution of thermal stresses in a truss using the superposition method

Variable	Problem I	Problem II	Final Solution
u_4	0	0	0
v_4	0	-0.4×10^{-3} m	-0.4×10^{-3} m
$P^{(1)}$	$-AE\alpha\Delta T^{(1)} = 0$	3,464 N	3,464 N
$P^{(2)}$	$-AE\alpha\Delta T^{(2)} = -10{,}000$ N	4,000 N	–6,000 N
$P^{(3)}$	$-AE\alpha\Delta T^{(3)} = 0$	3,464 N	3,464 N

The force resultants in the elements for problem II can be obtained from eq. (1.40). Substituting the element properties and displacements, we obtain:

$$P^{(1)} = 3{,}464\,\text{N},$$
$$P^{(2)} = 4{,}000\,\text{N},$$
$$P^{(3)} = 3{,}464\,\text{N}.$$

Then, the solution (displacements and forces) to the given problem is the sum of solutions to problems I and II as shown in table 1.6. Note that elements 1 and 3 are in tension, while element 2 is in compression.

If there were external forces acting at node 4 in the given problem, then they can be added to the fictitious forces in problem II. ■

1.5.2 Thermal Stresses Using FEA

In using the FEM for thermal stress problems, we combine the two problems in the previous subsection as one problem and solve for displacements and forces simultaneously. This procedure is similar to the superposition method. Consider the element equilibrium equation in eq. (1.27) for the uniaxial bar element. It states that the forces acting on an element are the product of element stiffness matrix and the vector of nodal displacements, that is, $\{\bar{\mathbf{f}}\} = [\bar{\mathbf{k}}]\{\bar{\mathbf{q}}\}$. This is similar to the linear elastic stress-strain relation at the element level, where stresses are a linear combination of strains at a point, $\{\sigma\} = [\mathbf{E}]\{\varepsilon\}$. However, we notice that in the presence of a temperature differential, the stresses are not a linear combination of strains, [see eq. (1.52)]. A similar adjustment has to be made at the element level equation also. In the presence of thermal stresses, eq. (1.32) can be modified as

$$\left\{\bar{\mathbf{f}}^{(e)}\right\} = \left[\bar{\mathbf{k}}^{(e)}\right]\left\{\bar{\mathbf{q}}^{(e)}\right\} - \left\{\bar{\mathbf{f}}_T^{(e)}\right\}, \tag{1.55}$$

where the element thermal force vector $\left\{\bar{\mathbf{f}}_T^{(e)}\right\}$ in the local coordinate system is given by

$$\left\{\bar{\mathbf{f}}_T^{(e)}\right\} = AE\alpha\Delta T \begin{Bmatrix} -1 \\ 0 \\ +1 \\ 0 \end{Bmatrix} \begin{matrix} \bar{u}_i \\ \bar{v}_i \\ \bar{u}_j \\ \bar{v}_j \end{matrix}. \tag{1.56}$$

Note that the row addresses or the DOFs corresponding to each force are indicated next to the force vector. Multiplying both sides of eq. (1.55) by the transpose of the transformation matrix, $[\mathbf{T}]^{\mathrm{T}}$, and also using $\{\bar{\mathbf{q}}\} = [\mathbf{T}]\{\mathbf{q}\}$, we obtain

$$\{\mathbf{f}\} = [\mathbf{k}]\{\mathbf{q}\} - \{\mathbf{f}_T\}, \tag{1.57}$$

where $[\mathbf{k}]$ is the element stiffness matrix defined in eq. (1.36) and $\{\mathbf{f}_T\}$ is the thermal force vector in the global coordinates given by

$$\{\mathbf{f}_T\} = AE\alpha\Delta T \begin{Bmatrix} -l \\ -m \\ +l \\ +m \end{Bmatrix} \begin{matrix} u_i \\ v_i \\ u_j \\ v_j \end{matrix}. \tag{1.58}$$

The vector $\{\mathbf{f}_T\}$ has four rows, and its row addresses are in the same order as those of $[\mathbf{k}]$.

If eq. (1.58) is substituted in the nodal equilibrium equations, we will obtain the global equations at the structural level as

$$[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\} + \{\mathbf{F}_{Ts}\}, \tag{1.59}$$

where $\{\mathbf{F}_{Ts}\}$ is the thermal load vector, which is obtained by assembling $\{\mathbf{f}_T\}$ of various elements. It is clear from eq. (1.59) that the increase in temperature is equivalent to adding an additional force to the member. After striking out the rows and columns corresponding to zero DOFs, we obtain the global equations as

$$[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\} + \{\mathbf{F}_T\}. \tag{1.60}$$

The assembly of $\{\mathbf{F}_T\}$ is similar to that of the stiffness matrix. Equation (1.60) is solved to obtain the unknown displacements $\{\mathbf{Q}\}$. In order to find the forces in elements, one must use eq. (1.54).

EXAMPLE 1.9 *Thermal stresses in a plane truss*

Solve the thermal stress problem in example 1.8 using the finite element method.

SOLUTION The element stiffness matrices are already given in example 1.8. The thermal force vectors are written below. For convenience, the rows and columns corresponding to zero DOFs are deleted and only those corresponding to active DOFs are shown.

$$\left\{\mathbf{f}_T^{(1)}\right\} = AE\alpha\Delta T^{(1)}\begin{Bmatrix} 1/2 \\ -\sqrt{3}/2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}\begin{matrix} u_4 \\ v_4 \end{matrix},$$

$$\left\{\mathbf{f}_T^{(2)}\right\} = AE\alpha\Delta T^{(2)}\begin{Bmatrix} 0 \\ -1 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10{,}000 \end{Bmatrix}\begin{matrix} u_4 \\ v_4 \end{matrix}, \tag{1.61}$$

$$\left\{\mathbf{f}_T^{(3)}\right\} = AE\alpha\Delta T^{(3)}\begin{Bmatrix} 1/2 \\ -\sqrt{3}/2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \end{Bmatrix}\begin{matrix} u_4 \\ v_4 \end{matrix}$$

Note that there is no thermal force vector for elements 1 and 3 because they are at the reference temperature. The row addresses are shown next to the thermal force vector in eq. (1.61).

Assembling the element stiffness matrices, we obtain the global stiffness matrix $[\mathbf{K}]$, and assembling the element thermal force vectors $\{\mathbf{f}_T\}$, we obtain the global thermal force vector $\{\mathbf{F}_T\}$ as

$$\{\mathbf{F}_T\} = \begin{Bmatrix} 0 \\ -10{,}000 \end{Bmatrix}. \tag{1.62}$$

The solution for displacements is obtained using the global equations

$$[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\} + \{\mathbf{F}_T\}. \tag{1.63}$$

Since there are no external forces in the present problem, $\{\mathbf{F}\} = \{\mathbf{0}\}$. Hence the global equations are:

$$\frac{10^7}{4}\begin{bmatrix} 2 & 0 \\ 0 & 10 \end{bmatrix}\begin{Bmatrix} u_4 \\ v_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -10{,}000 \end{Bmatrix}. \tag{1.64}$$

The solution to the above equations is obtained as:

$$\boxed{\begin{aligned} u_4 &= 0, \\ v_4 &= -0.4 \times 10^{-3}\,\text{m}. \end{aligned}}$$

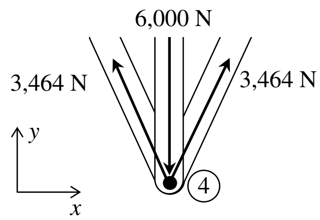


Figure 1.24 Force equilibrium at node 4

The force resultants in the elements are obtained from eq. (1.54):

$$\begin{aligned} P &= AE\left(\frac{\Delta L}{L} - \alpha\Delta T\right) \\ &= \frac{AE}{L}\left[l\left(u_j -\right) + m\left(v_j - v_i\right)\right] - AE\alpha\Delta T. \end{aligned} \tag{1.65}$$

Substituting the element properties and displacements, we obtain:

$$\boxed{\begin{aligned} P^{(1)} &= 3{,}464\text{N}, \\ P^{(2)} &= -6{,}000\text{N}, \\ P^{(3)} &= 3{,}464\text{N}. \end{aligned}}$$

The FE solution for displacements and forces above can be compared with those obtained from the superposition method presented in table 1.6.

One can check the force equilibrium at node 4. The three forces acting on node 4 are shown in figure 1.24. Summing the forces in the x and y directions,

$$\begin{aligned} \sum F_x &= -P^{(1)}\sin 30 + P^{(2)}\sin 30 = 0, \\ \sum F_y &= P^{(1)}\cos 30 - P^{(2)} + P^{(3)}\cos 30 \\ &= 3464 \times \frac{\sqrt{3}}{2} - 6000 + 3464 \times \frac{\sqrt{3}}{2} \\ &= 0. \end{aligned} \tag{1.66}$$

Thermal stress analysis of space trusses follows the same procedures. The thermal force vector $\{\mathbf{f}_T\}$ is a 6×1 matrix and is given by

$$\begin{array}{c} \qquad\qquad\qquad\quad u_i \quad v_i \quad w_i \quad u_j \quad v_j \quad w_j \\ \{\mathbf{f}_T\}^{\mathrm{T}} = AE\alpha\Delta T\{-l \;\; -m \;\; -n \;\; +l \;\; +m \;\; +n\}. \end{array} \tag{1.67}$$

In the above equation, $\{\mathbf{f}_T\}^{\mathrm{T}}$ is given as a row matrix with addresses shown above the elements of the matrix. Another difference between two- and three-dimensional thermal stress problems is in the calculation of force in an element. An equation similar to (1.65) for the three-dimensional truss element can be derived as

$$\begin{aligned} P &= AE\left(\frac{\Delta L}{L} - \alpha\Delta T\right) \\ &= \frac{AE}{L}\left[l\left(u_j - u_i\right) + m\left(v_j - v_i\right) + n\left(w_j - w_i\right)\right] - AE\alpha\Delta T, \end{aligned} \tag{1.68}$$

where l, m, and n are the direction cosines of the element. ■

EXAMPLE 1.10 *Thermal stresses in a space truss*

Use the FEM to determine the displacements and forces in the space truss, shown in figure 1.25. The coordinates of the nodes in meter units are given in table 1.7. The temperature of element 1 is raised by 100 °C above the reference

Table 1.7 Nodal coordinates of space truss structure in example 1.10

Node	x	y	z
1	0	0	0
2	0	–1	1
3	0	1	1
4	1	0	1
5	0	0	1

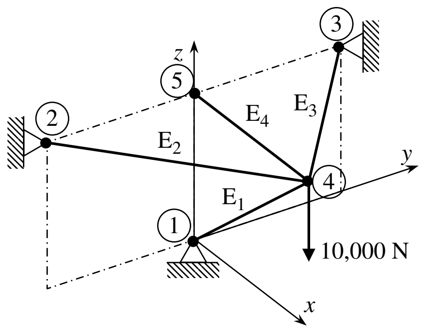


Figure 1.25 Three-bar space truss structure

temperature. Assume Young's modulus $E = 70$ Gpa and area of cross section $A = 1\ \text{cm}^2$. Assume CTE $\alpha = 20 \times 10^{-6}/$ °C. The magnitude of the downward force (negative z direction) at node 4 is equal to 10,000 N.

SOLUTION The first step is to determine the direction cosines of the elements. The length and direction cosines of each element are calculated using the formulas in eq. (1.47).

Element	LN1 (i)	LN2 (j)	L (m)	l	m	n
1	1	4	$\sqrt{2}$	$\sqrt{2}/2$	0	$\sqrt{2}/2$
2	2	4	$\sqrt{2}$	$\sqrt{2}/2$	$\sqrt{2}/2$	0
3	3	4	$\sqrt{2}$	$\sqrt{2}/2$	$-\sqrt{2}/2$	0
4	5	4	1	1	0	0

Stiffness matrices of elements 1 through 3 are the same as in example 1.6. The stiffness matrix of element 4 is as follows:

$$\left[\mathbf{k}^{(4)}\right] = 70 \times 10^5 \begin{bmatrix} 1 & 0 & 0 \\ 0 & 0 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_4 \\ v_4 \\ w_4 \end{matrix}.$$

We need to calculate the thermal force vector for element 1 only, as its temperature is different from the reference temperature. Using the formula in eq. (1.67) we obtain

$$\begin{matrix} & & u_1 & v_1 & w_1 & u_4 & v_4 & w_4 \\ \left\{\mathbf{f}_T^{(4)}\right\}^{\mathrm{T}} = 7000\sqrt{2}\{ & & -1 & 0 & -1 & 1 & 0 & 1\}. \end{matrix} \tag{1.69}$$

After assembly, we obtain the global equations in the form $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\} + \{\mathbf{F_T}\}$:

$$35\sqrt{2} \times 10^5 \begin{bmatrix} 1.5 + \sqrt{2} & 0 & 0.5 \\ 0 & 1.0 & 0 \\ 0.5 & 0 & 0.5 \end{bmatrix} \begin{Bmatrix} u_4 \\ v_4 \\ w_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ -10{,}000 \end{Bmatrix} + \begin{Bmatrix} 9900 \\ 0 \\ 9900 \end{Bmatrix}.$$

Solving the above equation, the unknown nodal displacements are obtained as

$$u_4 = 0.8368 \times 10^{-3}\,\text{m},$$
$$v_4 = 0,$$
$$w_4 = -0.8772 \times 10^{-3}\,\text{m}.$$

The forces in the elements can be calculated using eq. (1.68), and they are as follows:

$$P^{(1)} = -14{,}141\,\text{N},$$
$$P^{(2)} = +2{,}929\,\text{N},$$
$$P^{(3)} = +2{,}929\,\text{N},$$
$$P^{(4)} = +5{,}858\,\text{N}.$$

One can verify that the force equilibrium is satisfied at node 4. ■

1.6 FINITE ELEMENT MODELING PRACTICE FOR TRUSS

In this section, several analysis problems are used to discuss modeling issues as well as verifying the accuracy of analysis results with that of literature. The examples are presented in such a way that any finite element analysis program can be used to solve the problems. However, the analysis results can slightly be different because of implementation details of different finite element analysis programs.

1.6.1 Reaction Force of a Statically Indeterminate Bar[5]

A vertical prismatic bar shown in figure 1.26 is fixed at both ends. When two downward forces, $F_1 = 1000$ lb and $F_2 = 500$ lb, are applied as shown in the figure, calculate the reaction forces R_1 and R_2 at both ends. Use elastic modulus $E = 30 \times 10^6$ psi and a constant cross-sectional area $A = 0.1$ in^2.

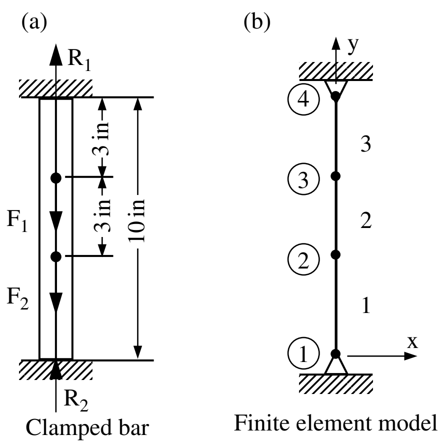


Figure 1.26 Statically indeterminate vertical bar

[5] Timoshenko, S. 1955. *Strength of Material, Part 1, Elementary Theory and Problems*, 3rd Edition. D. Van Nostrand Co., Inc., New York, NY. (Page 26, problem 10.)

Since the total applied load is 1500 lb, it is obvious that the sum of reaction forces will be 1500 lb. However, the individual reaction forces cannot be calculated from force equilibrium, as the structure is statically indeterminate. It is necessary to take into account compatibility in deformation to solve statically indeterminate systems. One way of solving the problem is by assuming that the bar is fixed only from the top and a force R_2 is applied at the bottom. Then, the unknown force R_2 can be calculated from the compatibility condition that the deflection at the bottom is zero. The deflection at the bottom due to F_1, F_2, and R_2 can be written as

$$\delta = \frac{(F_1 + F_2 - R_2)\times 3}{EA} + \frac{(F_2 - R_2)\times 3}{EA} - \frac{R_2 \times 4}{EA} = 0.$$

Note that each section of the bar is under different loads, which can easily be obtained from a free-body diagram. The reaction R_2 can be obtained by solving the above equation, $R_2 = 600$ lb. From the static equilibrium, the remaining reaction force $R_1 = 900$ lb can also be obtained.

One of the important advantages in finite element analysis is that there is no need to acknowledge the difference between statically determinate and indeterminate systems because the finite element analysis procedure automatically takes into account deformation. Figure 1.26(b) shows an example for finite element model for the statically indeterminate bar. Nodes 2 and 3 are located in order to apply the two nodes. It is possible to make more elements, but for this particular problem, the three elements will yield an accurate solution.

Even if all elements are vertically located, it is possible to model the bar using either 1D bar or 2D plane truss elements because deformation is limited along the axial direction. When 1D bar elements are used, the vertical direction is considered as the x coordinate, and forces are applied along the coordinate. In such a case, each node has a single DOF, u_I, and the total matrix size of the problem becomes 4×4. The assembled matrix equation becomes

$$10^6 \begin{bmatrix} 0.75 & -0.75 & 0 & 0 \\ -0.75 & 1.75 & -1 & 0 \\ 0 & -1 & 2 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} R_2 \\ -F_2 \\ -F_1 \\ R_1 \end{Bmatrix}. \tag{1.70}$$

In the above equation, the applied forces and reaction forces in figure 1.26 are used with positive being the positive y coordinate direction. Since nodes 1 and 4 are fixed, the first and fourth columns and rows are deleted in the process of applying the boundary conditions. Therefore, only a 2×2 matrix needs to be solved.

$$10^6 \begin{bmatrix} 1.75 & -1 \\ -1 & 2 \end{bmatrix} \begin{Bmatrix} u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} -500 \\ -1000 \end{Bmatrix}.$$

The above matrix equation can be solved for nodal displacement $u_2 = -8 \times 10^{-4}$ in. and $u_3 = -9 \times 10^{-4}$ in. The calculated nodal displacement can be substituted into eq. (1.70) to calculated reaction forces. From the first and fourth rows of eq. (1.70), we have

$$R_2 = 10^6 \times (0.75 \times u_1 - 0.75 \times u_2) = 600 \text{lb},$$
$$R_1 = 10^6 \times (-1 \times u_3 + 1 \times u_4) = 900 \text{lb}.$$

Note that the above reaction forces are identical to the previous analytical calculation.

Many finite element analysis programs do not have 1D bar elements. Instead, they use 2D plane truss or 3D space truss elements. For example, ANSYS has LINK180, which supports uniaxial tension-compression with three DOFs at each node: translations in the nodal x, y, and z directions, UX, UY, and YZ. On the other hand, Abaqus support T2D2 element for the two-dimensional plane truss.

When the above problem is solved using 3D space truss element, all elements are located in the y-coordinate direction and all forces are applied in the same direction. For boundary conditions, it is necessary to fix all three DOFs at nodes 1 and 4. On the other hand, it is necessary to fix UX and UZ for nodes 2 and 3 so that the motion is limited to the y direction.

1.6.2 Thermally Loaded Support Structure[6]

Two copper wires and a steel wire are connected by a rigid body as shown in figure 1.27. The three wires are all of an equal length of 20 in. and in the same cross-sectional area of $A = 0.1$ in^2. Initially, the structure was at a temperature of 70 °F. When a load $Q = 4000$ lb is applied and the temperature is increased to 80 °F simultaneously, find the stresses in the copper and steel wires. Note that the load Q is applied at the center of the rigid body. For the copper wire, use the elastic modulus $E_c = 1.6 \times 10^7$ psi and the thermal expansion coefficient $\alpha_c = 9.2 \times 10^{-6}$ in/in°F. For the steel wire, use the elastic modulus $E_s = 3.0 \times 10^7$ psi and the thermal expansion coefficient $\alpha_s = 7.0 \times 10^{-6}$ in/in°F.

Since the two copper wires have identical properties and the load is applied at the center of the rigid body, there will be no rotation, and all three wires will have the same amount of displacement. Therefore, the problem can be simplified into three 1D bars. The problem is statically indeterminate because not only we cannot calculate the internal load distribution between copper and steel from force equilibrium, but also the different thermal expansion coefficients cause different elongations.

1. ***Equilibrium under temperature change:*** In order to solve the problem analytically, we can use the method of superposition. That is, the stress caused by temperature change and the stress caused by the load can be calculated separately and then added to yield the final state of stresses. First, we ignore the rigid body for a moment and the three wires are free. When the temperature is increased by 10 °F, the elongation of copper and steel can be calculated by

$$\delta_c^1 = \alpha_c \Delta T L = 9.2 \times 10^{-6} \times 10 \times 20 = 0.00184 \text{ in},$$
$$\delta_s^1 = \alpha_s \Delta T L = 7.0 \times 10^{-6} \times 10 \times 20 = 0.00140 \text{ in}.$$

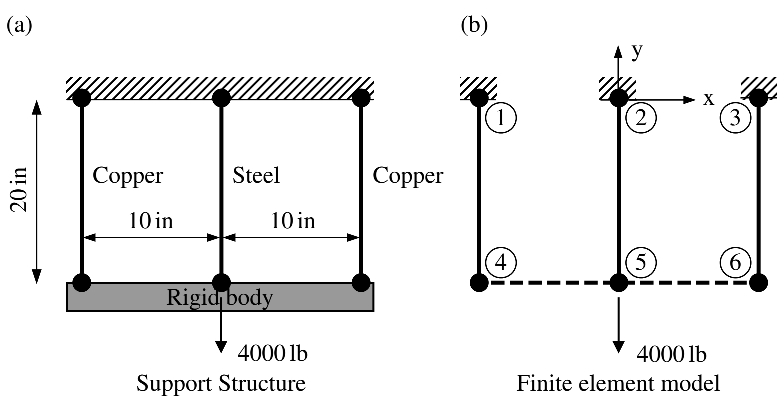


Figure 1.27 Thermally loaded three bars

[6] Timoshenko, S. 1955. *Strength of Material, Part 1, Elementary Theory and Problems*, 3rd Edition. D. Van Nostrand Co., Inc., New York, NY. (Page 30, problem 9.)

The superscript 1 stands for the stage when the temperature changes but the load Q is not applied. At this stage, there is no stress in the wires as they are not constrained. Connecting them with a rigid body means that the steel needs to be elongated further while the coppers need to be compressed. Let us assume that δ^1 is the final displacement after the rigid body is located. Then the internal forces can be written in terms of δ^1 as

$$F_c^1 = \frac{E_c A}{L}\left(\delta^1 - \delta_c^1\right),$$
$$F_s^1 = \frac{E_s A}{L}\left(\delta^1 - \delta_s^1\right). \tag{1.71}$$

The unknown displacement δ^1 can be calculated from the condition that there is no externally applied force. That is, the sum of internal forces must vanish.

$$2F_c^1 + F_s^1 = 2\frac{E_c A}{L}\left(\delta^1 - \delta_c^1\right) + \frac{E_s A}{L}\left(\delta^1 - \delta_s^1\right) = 0.$$

In the above equation, F_c is multiplied by two because there are two copper wires. The above equation can be solved for $\delta^1 = 0.00163$ in. The internal forces and stresses can be calculated by substituting δ^1 into eq. (1.71) as

$$F_c^1 = \frac{E_c A}{L}\left(\delta^1 - \delta_c^1\right) = -17.03\text{lb} \qquad \sigma_c^1 = \frac{F_c^1}{A} = -170.3\text{psi},$$
$$F_s^1 = \frac{E_s A}{L}\left(\delta^1 - \delta_s^1\right) = 34.06\text{lb} \qquad \sigma_s^1 = \frac{F_s^1}{A} = 340.6\text{psi}.$$

2. ***Equilibrium under applied load Q:*** When $Q = 4000$ lb is applied at the center of the rigid body, all three wires will displace in the same amount. When the vertical displacement is δ^2, the sum of internal forces must be in equilibrium with the applied load as

$$\left(2 \times \frac{E_c A}{L} + \frac{E_s A}{L}\right) \times \delta^2 = 4000,$$

from which the displacement $\delta^1 = 0.0123$ in. can be obtained. Then, the internal forces and stresses can be calculated by

$$F_c^2 = \frac{E_c A}{L}\delta^2 = 1032.3\text{lb} \qquad \sigma_c^2 = \frac{F_c^2}{A} = 10323\text{psi},$$
$$F_s^2 = \frac{E_s A}{L}\delta^2 = 1935.5\text{lb} \qquad \sigma_s^2 = \frac{F_s^2}{A} = 19355\text{psi}.$$

3. ***Combined internal forces and stresses:*** An important characteristic of a linear system is that the stresses from different loads can be superposed together to yield the stress at the combined loads. The final results become

$$\delta = \delta^1 + \delta^2 = 0.01453 \text{ in},$$
$$F_c = F_c^1 + F_c^2 = 1015.2 \text{ lb} \quad \sigma_c = \sigma_c^1 + \sigma_c^2 = 10152 \text{ psi},$$
$$F_s = F_s^1 + F_s^2 = 1969.5 \text{ lb} \quad \sigma_s = \sigma_s^1 + \sigma_s^2 = 19695 \text{ psi}.$$

4. ***Finite element modeling and analysis:*** Figure 1.27(b) shows a finite element model, where three bar elements are used to represent the three wires. Similar to statically indeterminate systems, it is unnecessary to separate the temperature change from the applied load. The temperature change can be considered as a thermal load as shown in eq. (1.55). For simplicity, 1D bar

elements are used to model the thermally loaded support structure. In order to do that, the three element matrix equations with the thermal load can be written as

$$\text{Element 1: } \frac{E_cA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_4 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_4^{(1)} \end{Bmatrix} + AE_c\alpha_c\Delta T\begin{Bmatrix} -1 \\ 1 \end{Bmatrix}.$$

$$\text{Element 2: } \frac{E_sA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_2 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix} + AE_s\alpha_s\Delta T\begin{Bmatrix} -1 \\ 1 \end{Bmatrix}.$$

$$\text{Element 3: } \frac{E_cA}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_3 \\ u_6 \end{Bmatrix} = \begin{Bmatrix} f_3^{(3)} \\ f_6^{(3)} \end{Bmatrix} + AE_c\alpha_c\Delta T\begin{Bmatrix} -1 \\ 1 \end{Bmatrix}.$$

The assembly of the above three element matrix equations yields a 6×6 structural matrix equation. Since nodes 1, 2, and 3 are fixed, the first three rows and columns are deleted in the process of applying boundary conditions. Therefore, after applying boundary conditions, the following global matrix equation can be obtained:

$$\begin{bmatrix} 80000 & 0 & 0 \\ 0 & 150000 & 0 \\ 0 & 0 & 80000 \end{bmatrix}\begin{Bmatrix} u_4 \\ u_5 \\ u_6 \end{Bmatrix} = \begin{Bmatrix} F_4 \\ F_5 \\ F_6 \end{Bmatrix} + \begin{Bmatrix} 147.2 \\ 210 \\ 147.2 \end{Bmatrix}.$$

The condition of a rigid body can be written as $u_4 = u_5 = u_6$. There are many different ways of considering the effect of a rigid body; a simple method can be adding three equations together, which yields the following scalar equation:

$$(80000 + 150000 + 80000) \times u_5 = (F_4 + F_5 + F_6) + (147.2 + 210 + 147.2).$$

Considering the fact that $F_4 + F_5 + F_6 = Q = 4000$ lb., the above equation can be solved for $u_5 = 0.01453$ in. $= u_4 = u_6$. Now the calculated displacement can be substituted into the element matrix equation to calculate the element forces as

$$P^{(1)} = f_j^{(1)} = \frac{E_cA}{L}(u_4 - u_1) - AE_c\alpha_c\Delta T = 1015.2 \text{ lb},$$

$$P^{(2)} = f_j^{(2)} = \frac{E_sA}{L}(u_5 - u_1) - AE_s\alpha_s\Delta T = 1969.5 \text{ lb},$$

$$P^{(3)} = f_j^{(3)} = \frac{E_cA}{L}(u_6 - u_1) - AE_c\alpha_c\Delta T = 1015.2 \text{ lb}.$$

Note that the element forces are identical to the internal forces in the analytical calculation. Therefore, the stresses will also be identical.

1.6.3 Deflection of a Two-Bar Truss[7]

A structure consisting of two equal steel bars, each of length $L = 15$ ft. and cross-sectional area $A = 0.5$ in.^{2}, with hinged ends, is subjected to the action of a load $F = 5000$ lb. Determine the stress, σ, in the bars and the vertical deflection, δ, at node 2. Neglect the weight of the bars as a small quantity in comparison with the load F. For material property, use $E = 30 \times 10^6$ psi.

[7] Timoshenko, S. 1955. *Strength of Material, Part 1, Elementary Theory and Problems*, 3rd Edition. D. Van Nostrand Co., Inc., New York, NY. (Page 30, problem 9.)

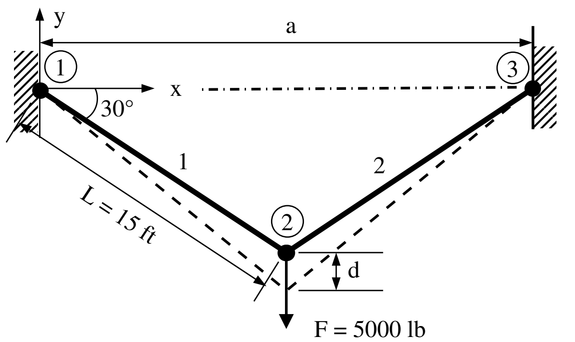


Figure 1.28 Two-bar truss

Since the truss is a two-force member, it can only support the force in the axial direction. In addition, due to symmetry, it is obvious that the force in member 1 will be the same as that of member 2. The sum of vertical components of the two member forces is in equilibrium with the externally applied load. That is,

$$2f\sin 30 = 5000 \quad \Rightarrow \quad f = 5000 \text{ lb.}$$

Therefore, the stress in both members are

$$\sigma^{(1)} = \sigma^{(2)} = \frac{f}{A} = 10{,}000 \text{ psi.}$$

The internal forces can cause elongation of the truss, which can be calculated by

$$\delta' = \frac{fL}{EA} = \frac{5000 \times 15 \times 12}{3 \times 10^7 \times 0.5} = 0.06 \text{ in.}$$

That is, the new length of the truss becomes 180.06 in. Using the geometry in figure 1.28, the new angle after deformation can be calculated by

$$\theta = \cos^{-1}\left(\frac{180 \times \cos(30)}{180.06}\right) = 30.033°.$$

Therefore, the vertical displacement, δ, in figure 1.28 can be calculated by

$$\delta = 180.06 \times \sin(30.033) - 180 \times \sin(30) = 0.12 \text{ in.}$$

For finite element modeling, two equal-length, 2D plane truss finite element can be used as shown in figure 1.28. In the case of plane truss elements, the following table can be used to define element connectivity and the element matrix equation:

Element	LN1 (i)	LN2 (j)	AE/L (N/m)	ϕ (degrees)	$l = \cos\phi$	$m = \sin\phi$
1	1	2	83333	–30	$\sqrt{3}/2$	–1/2
2	3	2	83333	–150	$-\sqrt{3}/2$	–1/2

In order to simplify the expression, the boundary conditions can be applied at the element level. That is, only free degrees of freedom are used in the expression. In this example, since nodes 1 and 3 are fixed, their degrees of freedom are removed at the element level. The two element matrix equations can be defined as

Element 1: $83333\begin{bmatrix} 3/4 & -\sqrt{3}/4 \\ -\sqrt{3}/4 & 1/4 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} f_{x2}^{(1)} \\ f_{y2}^{(1)} \end{Bmatrix}.$

Element 2: $83333\begin{bmatrix} 3/4 & \sqrt{3}/4 \\ \sqrt{3}/4 & 1/4 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} f_{x2}^{(2)} \\ f_{y2}^{(2)} \end{Bmatrix}.$

The assembled system of equation becomes

$$83333\begin{bmatrix} 3/2 & 0 \\ 0 & 1/2 \end{bmatrix}\begin{Bmatrix} u_2 \\ v_2 \end{Bmatrix} = \begin{Bmatrix} 0 \\ -5000 \end{Bmatrix}.$$

The above equation solves for $u_2 = 0$ and $v_2 = -0.12$ in. Note that the vertical deflection $\delta = -v_2$. In order to calculate element stress, the formula in eq. (1.40) is used for the element force as

$$P^{(1)} = \left(\frac{AE}{L}\right)^{(1)}\left(l\left(u_j - u_i\right) + m\left(v_j - v_i\right)\right) = 83333\left(-\frac{1}{2}(-0.12 - 0)\right) = 5000 \text{ lb},$$

and then, the element stress can be obtained by dividing the element force by the cross-sectional area:

$$\sigma^{(1)} = \frac{P^{(1)}}{A} = 10{,}000 \text{ psi}.$$

Since the truss is symmetric, element 2 will yield identical results.

1.7 PROJECTS

Project 1.1 Analysis and Design of a Space Truss

A space frame structure as shown in figure 1.29 consists of 25 truss members. Initially, all members have the same circular cross-sections with diameter $d = 2.0$ in. At nodes 1 and 2, a constant force $F = 60{,}000$ lb. is applied in the y direction. Four nodes (7, 8, 9, and 10) are fixed on the ground. The frame structure is made of a steel material whose properties are Young's modulus $E = 3 \times 10^7$ psi, Poisson's ratio $\nu = 0.3$, yield stress $\sigma_Y = 37{,}000$ psi, and density $\rho = 0.284$ lb./in^3. The safety factor $N = 1.5$ is used. Due to the manufacturing constraints, the diameter of the truss members should vary between 0.1 in. and 2.5 in.

1. Solve the initial truss structure using truss finite elements. Provide a plot that shows labels for elements and nodes along with boundary conditions. Provide deformed geometry of the structure and a table of stress in each element.
2. Minimize the structural weight by changing the cross-sectional diameter of each truss element, while all members should be safe under the given yield stress and the safety factor. You can use the symmetric geometry of the structure. Identify zero-force members. For zero-force members, use the lower bound of the cross-sectional diameter. Provide deformed geometry at the optimum design along with a table of stress at each element. Provide structural weights of initial and optimum designs.

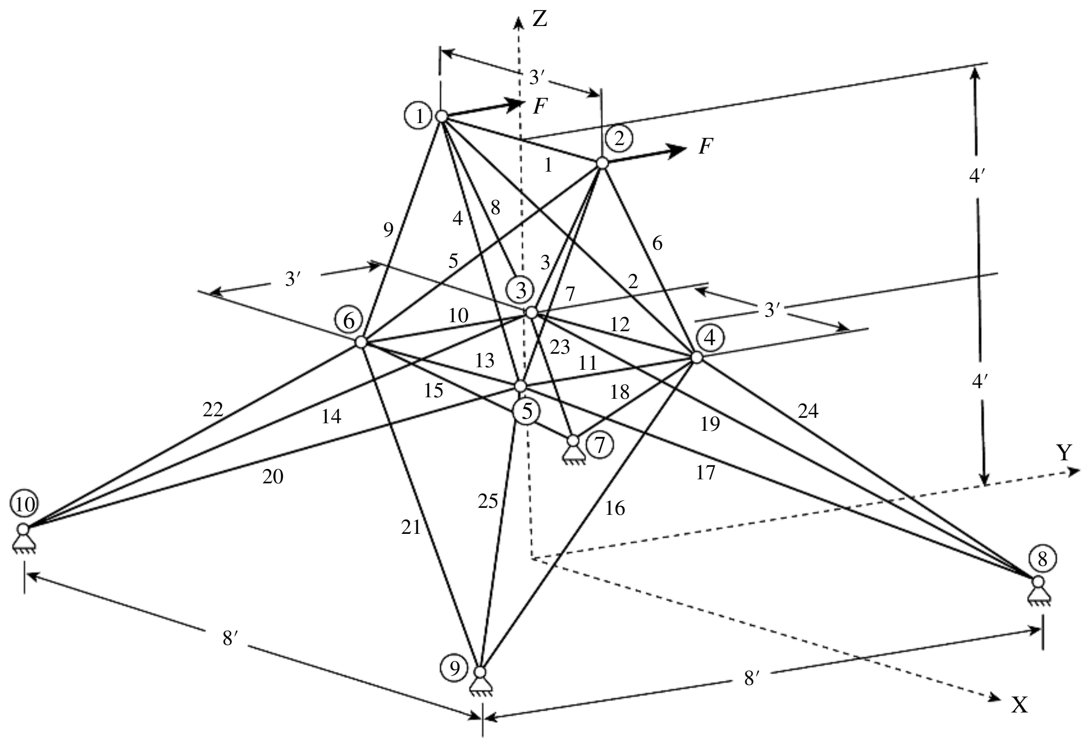


Figure 1.29 25–member space truss

Project 1.2 Analysis and Design of a Plane Truss 1

The truss shown in figure 1.30 has two elements. The members are made of the aluminum hollow square cross section. The outer dimension of the square is 12 mm, and the inner dimension is 9 mm. (The wall thickness is 1.5 mm on all four sides.) Assume Young's modulus $E = 70$ GPa, and yield strength $\sigma_Y = 70$ MPa. The magnitude of the force at node 1 (F) is equal to 1,000 N.

1. Use FEM to determine the displacements at node 1 and axial forces in elements 1 and 2. Use von Mises yield theory to determine if the elements will yield or not. Use Euler buckling load ($P_{\text{cr}} = \pi^2 EI/L^2$) to determine if the elements under compressive loads will buckle. In the above expression, P_{cr} is the axial compressive force, E is Young's modulus, I is the moment of inertia of the cross section given by $I = \left(a_o^4 - a_i^4\right)/12$, where a_o and a_i, respectively, are the outer and inner dimensions of the hollow square cross section, and L is the length of the element.
2. Redesign the truss so that both the stress and buckling constraints are satisfied with a safety factor of N not less than 2 for stresses, and N not less than 1.2 for buckling. Your design goal should be to reduce the weight of the truss as much as possible. The truss should be contained within the virtual rectangle shown by the dashed lines. Node 1 must be present to take the downward load $F =$ 1,000 N. The nodes at the left wall have to be fixed completely. Nodes not attached to the wall have to be completely free to move in the x and y directions. Use the same cross section for all elements. Calculate the mass of the truss you have designed. Assume the density of aluminum as 2,800 kg/m^3.Draw the truss you have designed and provide the nodal coordinates and element connectivity in the form of a table. Results should also include the nodal displacements, forces in each member, and the safety factors for stresses and buckling for each element in the form of tables.

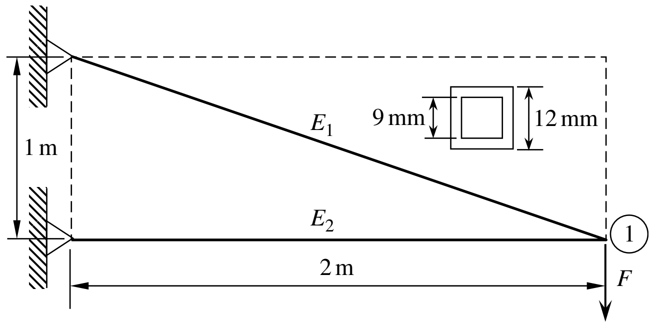


Figure 1.30 Plane truss and design domain for Project 1.2

Project 1.3 Analysis and Design of a Plane Truss 2

Consider a plane truss in figure 1.31. The horizontal and vertical members have length l, while inclined members have length $\sqrt{2}l$. Assume Young's modulus $E = 100$ GPa, cross-sectional area $A = 1.0\,\text{cm}^2$, and $l = 0.3$ m.

1. Use an FE program to determine the deflections and element forces for the following three load cases. Present you results in the form of a table.
   Load Case A) $F_{x13} = F_{x14} = 10{,}000$ N
   Load Case B) $F_{y13} = F_{y14} = 10{,}000$ N
   Load Case C) $F_{x13} = 10{,}000$ N and $F_{x14} = -10{,}000$ N
2. Assuming that the truss behaves like a cantilever beam, one can determine the equivalent cross-sectional properties of the beam from the results for cases A through C above. The three beam properties are axial rigidity $(EA)_{eq}$ (this is *different* from the AE of the truss member), flexural rigidity $(EI)_{eq}$, and shear rigidity $(GA)_{eq}$. Let the beam length be equal to L ($L = 6 \times 0.3 = 1.8$ m). The axial deflection of a beam due to an axial force F is given by:

$$u_{tip} = \frac{FL}{(EA)_{eq}}. \tag{1.72}$$

   The transverse deflection due to a transverse force F at the tip is:

$$v_{tip} = \frac{FL^3}{3(EI)_{eq}} + \frac{FL}{(GA)_{eq}}. \tag{1.73}$$

   In eq. (1.73) the first term on the RHS represents the deflection due to flexure and the second term, due to shear deformation. In the elementary beam theory (Euler-Bernoulli beam theory) we neglect the shear deformation, as it is usually much smaller than the flexural deflection.
   The transverse deflection due to an end couple C is given by:

$$v_{tip} = \frac{CL^2}{2(EI)_{eq}}. \tag{1.74}$$

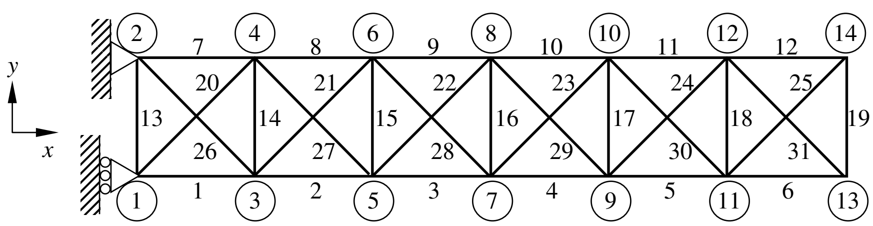


Figure 1.31 Plane truss and design domain for Project 1.3

Substitute the average tip deflections obtained in part 1 in eqs. (1.72)–(1.74) to compute the equivalent section properties: $(EA)_{eq}$, $(EI)_{eq}$, and $(GA)_{eq}$.
You may use the average of deflections at nodes 13 and 14 to determine the equivalent beam deflections.

3. Verify the beam model by adding two more bays to the truss ($L = 8 \times 0.3 = 2.4$ m). Compute the tip deflections of the extended truss for the three load cases A–C using the FE program. Compare the FE results with deflections obtained from the equivalent beam model (eqs. (1.72)–(1.74)).

Project 1.4 Fully Stressed Design of a Ten-Bar Truss

The fully stressed design is often used for truss structures. The idea is that we should remove material from members that are not fully stressed unless prevented by minimum cross-sectional area constraint. Practically, at every design cycle, the new cross-sectional area can be found using the following relation:

$$A_{\text{new}}^{(e)} = \frac{\sigma_{\text{old}}^{(e)}}{\sigma_{\text{allowable}}^{(e)}} A_{\text{old}}^{(e)}.$$

A ten-bar truss structure shown in figure 1.32 is under two loads, P_1 and P_2. The design goal is to minimize the weight, W, by varying the cross-sectional areas, A_i, of the truss members. The stress of the member should be less than the allowable stress with the safety factor. For manufacturing reasons, the cross-sectional areas should be greater than the minimum value. Input data are summarized in the table. Find optimum design using a fully stressed design.

Parameters	Values
Dimension, b	360 inches
Safety factor, S_F	1.5
Load, P_1	66.67 kips
Load, P_2	66.67 kips
Density, ρ	0.1 lb/in^3
Modulus of elasticity, E	10^4 ksi
Allowable stress, $\sigma_{\text{allowable}}$	25 ksi*
Initial area A_i	1.0 in^2
Minimum cross-sectional area	0.1 in^2

* for Element 9, allowable stress is 75 ksi

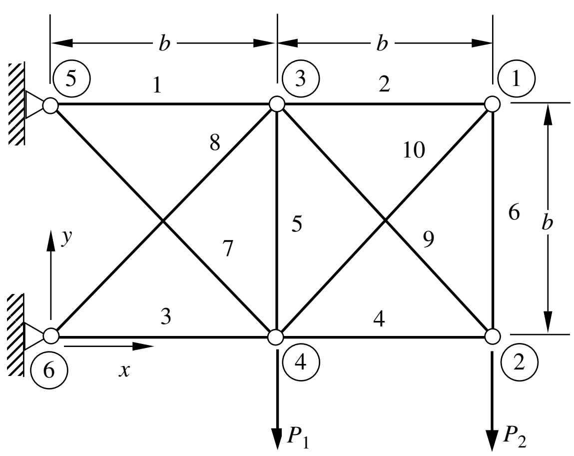


Figure 1.32 Ten-bar truss structure for project 1.4

1.8 EXERCISES

1. Answer the following descriptive questions

(a) Write five properties of the element stiffness matrix.

(b) Write five properties of the structural stiffness matrix.

(c) Write five properties of the global stiffness matrix.

(d) Will subdividing a truss element into many smaller elements improve the accuracy of the solution? Explain.

(e) Explain when the element force $P^{(e)}$ is positive or negative.

(f) For a given spring element, explain why we cannot calculate nodal displacement u_i and u_j when nodal forces $f_i^{(e)}$ and $f_j^{(e)}$ are given.

(g) Explain what "striking the rows" and "striking the columns" are.

(h) Once the gobal DOFs {**Q**} is solved, explain two methods of calculating nodal reaction forces.

(i) Explain why we need to define the local coordinate in 2D truss element.

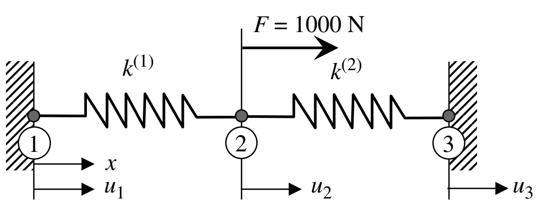
(j) Explain how to determine the angle ϕ of 2D truss element.

(k) When the connectivity of an element is changed from i → j to j → i, will the global stiffness matrix be the same or different? Explain why.

(l) For the bar with fixed two ends in figure 1.22, when temperature is increased by ΔT, which of the following strains are not zero? Total strain, mechanical strain, or thermal strain?

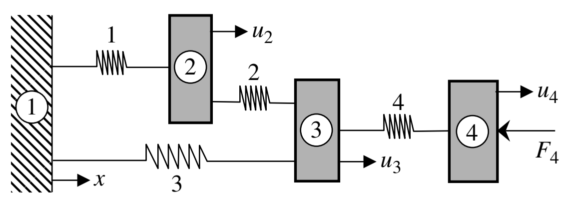
(m) Describe the finite element model you would use for a thin slender bar pinned at both ends with a transverse (perpendicular to the bar) concentrated load applied at the middle. (1) Draw a figure to show the elements, loads, and boundary conditions. (2) What type of elements will you use?

2. Calculate the displacement at node 2 and reaction forces at nodes 1 and 3 of the springs shown in the figure using two spring elements. A force $F = 1000$ N is applied at node 2. Use $k^{(1)} = 2000$ N/mm and $k^{(2)} = 3000$ N/mm.

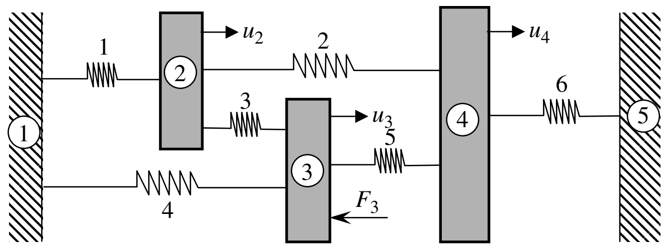


3. Repeat problem 2 by changing node numbers; that is, node 3 is now node 1, and node 1 is now node 3. Check if the results are the same as those of problem 2.

4. Three rigid bodies, 2, 3, and 4, are connected by four springs as shown in the figure. A horizontal force of 1,000 N is applied on body 4 as shown in the figure. Find the displacements of the three bodies and the forces (tensile/compressive) in the springs. What is the reaction at the wall? Assume the bodies can undergo only translation in the horizontal direction. The spring constants (N/mm) are: $k^{(1)} = 400$, $k^{(2)} = 500$, $k^{(3)} = 400$, $k^{(4)} = 200$.



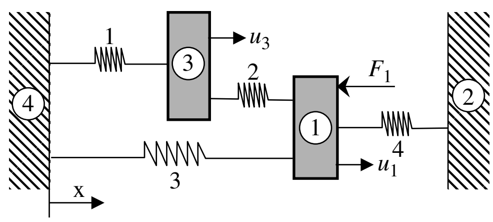
5. Three rigid bodies, 2, 3, and 4, are connected by six springs as shown in the figure. The rigid walls are represented by 1 and 5. A horizontal force $F_3 = 2000$ N is applied on body 3 in the direction shown in the figure. Find the displacements of the three bodies and the forces (tensile/compressive) in the springs. What are the reactions at the walls? Assume the bodies can undergo only translation in the horizontal direction. The spring constants (N/mm) are: $k^{(1)} = 200$, $k^{(2)} = 400$, $k^{(3)} = 600$, $k^{(4)} = 300$, $k^{(5)} = 500$, $k^{(6)} = 300$.



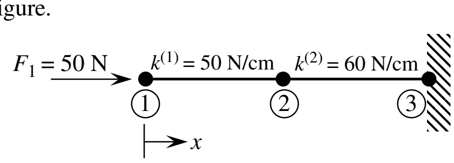
6. Consider the spring-rigid body system described in problem 5. What force F_2 should be applied on body 2 in order to keep it from moving? How will this affect the support reactions?

   Hint: Impose the boundary condition $u_2 = 0$ in the finite element model and solve for displacements u_3 and u_4. Then, the force F_2 will be the reaction at node 2.

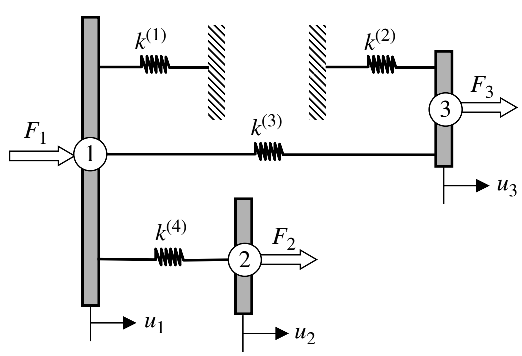
7. Four rigid bodies, 1, 2, 3, and 4, are connected by four springs as shown in the figure. A horizontal force of 1,000 N is applied on body 1 as shown in the figure. Using FE analysis, (a) find the displacements of bodies 1 and 3, (2) find the element force (tensile/compressive) of spring 1, and (3) find the reaction force at the right wall (body 2). Assume the bodies can undergo only translation in the horizontal direction. The spring constants (N/mm) are: $k^{(1)} = 500$, $k^{(2)} = 300$, $k^{(3)} = 400$, and $k^{(4)} = 300$. Do not change node and element numbers.



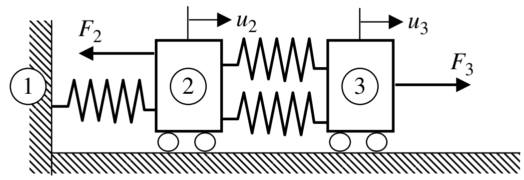
8. Determine the nodal displacements, element forces, and reaction forces using the direct stiffness method for the two-bar truss shown in the figure.



9. In the structure shown, rigid blocks are connected by linear springs. Imagine that only horizontal displacements are allowed. Write the global equilibrium equations $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$ after applying displacement boundary conditions in terms of spring stiffness $k^{(i)}$, displacement DOFs u_i, and applied loads F_i.



10. The spring-mass system shown in the figure is in equilibrium under the applied loads. The element connectivity table shows node numbers associated with each element and the stiffness of each element.

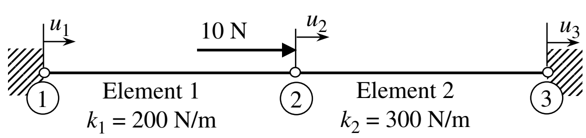


Element	Local Node 1	Local Node 2	Stiffness, k_i
1	1	2	2000 N/m
2	2	3	1000 N/m
3	2	3	500 N/m

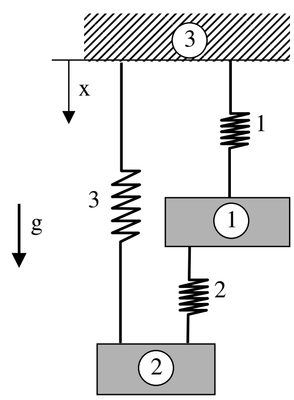
(a) Write the structural matrix equations $[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}$ by assembling the structural stiffness matrix $[\mathbf{K}_s]$ and the force vector $\{\mathbf{F}_s\}$. Use $F_3 = 5$ N and $F_2 = 2$ N.

(b) Apply the boundary conditions and solve for the displacement of the two masses.

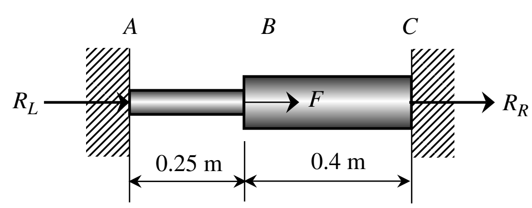
11. A structure is composed of two one-dimensional bar elements. When a 10 N force is applied to node 2, calculate the displacement vector $\{\mathbf{Q}\}^T = \{u_1, u_2, u_3\}$ using the finite element method.



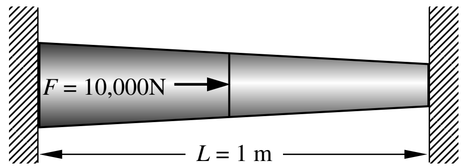
12. Two rigid masses, 1 and 2, are connected by three springs as shown in the figure. When gravity is applied with $g = 9.85$ m/sec^2, using FE analysis, (a) find the displacements of masses 1 and 2, (2) find the element forces of all the springs, and (3) find the reaction force at the wall (body 3). Assume the bodies can move only in the vertical direction and cannot rotate. The spring constants (N/mm) are $k^{(1)} = 400$, $k^{(2)} = 500$, and $k^{(3)} = 500$. The masses (kg) are $m_1 = 20$ and $m_2 = 40$.



13. Use the finite element method to determine the axial force P in each portion, AB and BC, of the uniaxial bar. What are the support reactions? Assume: $E = 100$ GPa, the area of cross sections of the two portions AB and BC are, respectively, 10^{-4} m^2 and 2×10^{-4} m^2, and $F = 10{,}000$ N. The force F is applied at the cross section at B.



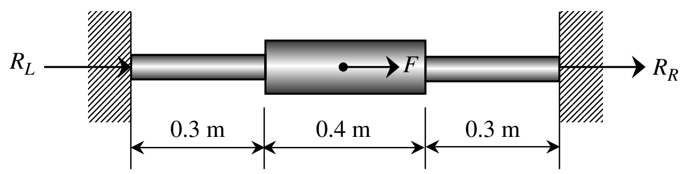
14. Consider a tapered bar of circular cross section. The length of the bar is 1 m, and the radius varies as $r(x) = 0.050 - 0.040x$, where r and x are in meters. Assume Young's modulus = 100 MPa. Both ends of the bar are fixed, and $F = 10{,}000$ N is applied at the center. Determine the displacements, axial force distribution, and the wall reactions using four elements of equal length.



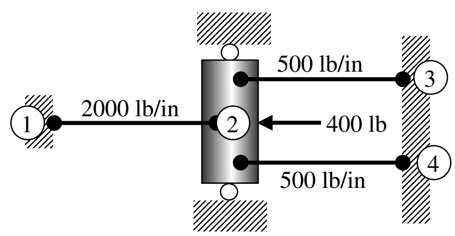
Hint: To approximate the area of cross section of a bar element, use the geometric mean of the end areas of the element, i.e., $A^{(e)} = \sqrt{A_i A_j} = \pi r_i r_j$.

15. The stepped bar shown in the figure is subjected to a force at the center. Use the finite element method to determine the displacement at the center and reactions R_L and R_R.

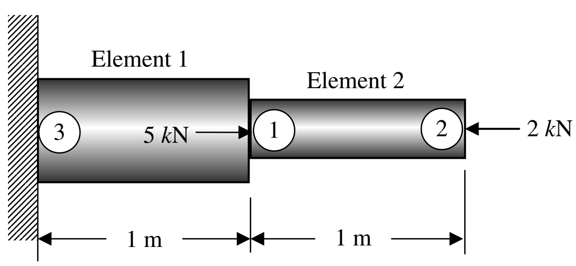
Assume: $E = 100$GPa, the area of cross sections of the three portions shown are, respectively, 10^{-4} m^2, 2×10^{-4} m^2, and 10^{-4} m^2, and $F = 10{,}000$ N.



16. Using the direct stiffness matrix method, find the nodal displacements and the forces in each element and the reactions.



17. A stepped bar is clamped at one end and subjected to concentrated forces as shown.
Note: the node numbers are not in usual order!



Assume: $E = 100$ GPa, small area of cross section = 1 cm^2, and large area of cross section = 2 cm^2.

(a) Write the element stiffness matrices of elements 1 and 2 showing the row addresses.

(b) Assemble the above element stiffness matrices to obtain the following structural level equations in the form $[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}$.

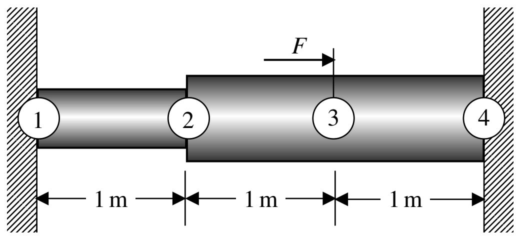
(c) Delete the rows and columns corresponding to zero DOFs to obtain the global equations in the form of $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$.

(d) Determine the displacements and element forces.

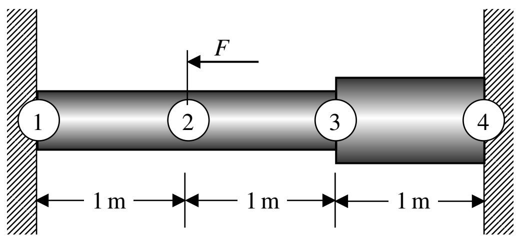
18. A stepped bar is clamped at both ends. A force of $F = 10{,}000$ N is applied as shown in the figure. Areas of cross sections of the two portions of the bar are 1×10^{-4} m^2 and 2×10^{-4} m^2, respectively. Young's modulus =10 GPa. Use three bar elements to solve the problem.

(a) Sketch the displacement field $u(x)$ as a function of x on an x-y graph.

(b) Calculate the stress at a distance of 2.5 m from the left support.



19. Repeat problem 18 for the stepped bar shown in the figure.



20. The finite element equation for the uniaxial bar can be used for other types of engineering problems if a proper analogy is applied. For example, consider the piping network shown in the figure. Each section of the network can be modeled using an FE. If the flow is laminar and steady, we can write the equations for a single pipe element as:

$$q_i = K\left(P_i - P_j\right)$$
$$q_j = K\left(P_j - P_i\right)$$

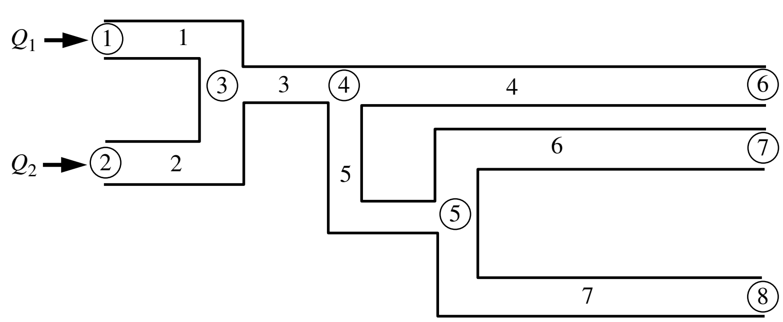
where q_i and q_j are fluid flow at nodes i and j, respectively; P_i and P_j are fluid pressure at nodes i and j, respectively; and K is

$$K = \frac{\pi D^4}{128\mu L},$$

where D is the diameter of the pipe, μ is the viscosity, and L is the length of the pipe. The fluid flow is considered positive away from the node. The viscosity of the fluid is 9×10^{-4} Pa·s.

(a) Write the element matrix equation for the flow in the pipe element.

(b) The net flow rates into nodes 1 and 2 are 10 and 15 m^3/s, respectively. The pressures at the nodes 6, 7, and 8 are all zero. The net flow rate into the nodes 3, 4, and 5 are all zero. What is the outflow rate for elements 4, 6, and 7?



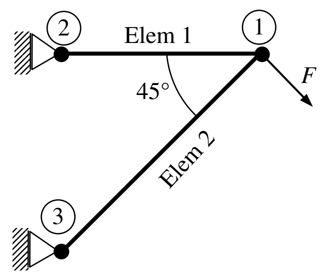
Elem	1	2	3	4	5	6	7
D(mm)	40	40	50	25	40	25	25
L(m)	1	1	1	4	2	3	3

21. The truss structure shown in the figure supports a force F. The finite element method is used to analyze this structure using two truss elements as shown in the figure. The cross-sectional area for both elements is $A = 2\ \text{in}^2$, and the lengths are $L^{(2)} = 10$ ft. Young's Modulus of the material $E = 30 \times 10^6$ psi.

(a) Compute the transformation matrix [**T**] for element 2 that enables you to transform between global and local coordinates (as shown in equation below).

$$\begin{Bmatrix} \bar{u}_1 \\ \bar{u}_3 \end{Bmatrix} = [\mathbf{T}] \begin{Bmatrix} u_1 \\ v_1 \\ u_3 \\ v_3 \end{Bmatrix}.$$

(b) It is determined after solving the final equations that the displacement components of the node 1 are: $u_1 = 0.5 \times 10^{-2}$ in. and $v_1 = -1.5 \times 10^{-2}$ in. What are the strain, stress, and force in element 2?



22. The properties of the two elements of a plane truss are given in the table below. Note that an external force of 10,000 N is acting on the truss at node 2.

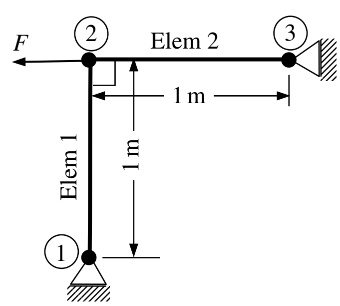
Elem.	$i \to j$	ϕ	l	m	L (m)	A (cm^2)	E (GPa)	α (/°C)	ΔT (°C)
1	$1 \to 2$	90	0	1	1	1	100	20×10^{-6}	−100
2	$2 \to 3$	0	1	0	1	1	100	20×10^{-6}	0

(a) Write the thermal force vector for each element. Indicate row addresses clearly.

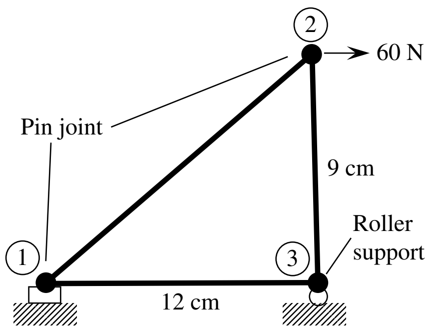
(b) Assemble the thermal force vectors to form the global thermal force $\{\mathbf{F}_T\}$, which is a 2×1 matrix.

(c) The problem was solved using FEA to obtain the displacements as $u_2 = -1$ mm, $v_2 = -2$ mm. Determine the element force P in each element.

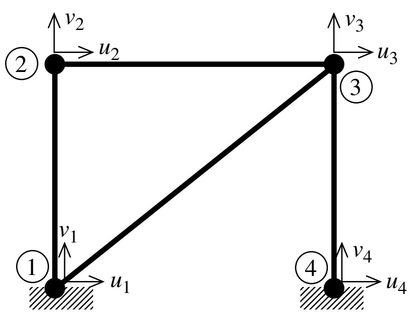
(d) Show that equilibrium is satisfied at node 2.



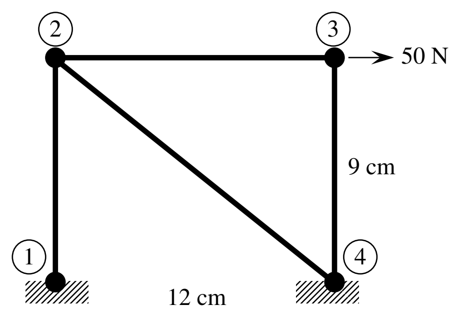
23. For a two-dimensional truss structure as shown in the figure, determine displacements of the nodes and normal stresses developed in the members using the direct stiffness method. Use $E = 30 \times 10^6$ N/cm^2, and the diameter of the circular cross-section is 0.25 cm.



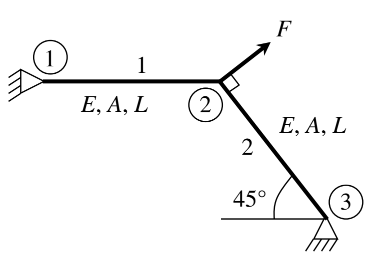
24. The 2D truss shown in the figure is assembled to build the global matrix equation. Before applying boundary conditions, the dimension of the global stiffness matrix is 8×8. Write (row, column) indices of locations corresponding to nonzero DOFs of the stiffness matrix, such as (1,1), (1,2), and so forth. The order of global DOFs is $\{\mathbf{Q}\} = \{u_1, v_1, u_2, v_2, u_3, v_3, u_4, v_4\}^T$. Since the stiffness matrix is symmetric, only write its upper-right portion.



25. For a two-dimensional truss structure as shown in the figure, determine displacements of the nodes and normal stresses developed in the members using a commercial finite element analysis program. Use $E = 30 \times 10^6$ N/cm^2, and the diameter of the circular cross-section is 0.25 cm.



26. The truss shown in the figure supports force F at node 2. The finite element method is used to analyze this structure using two truss elements as shown.



(a) Compute the transformation matrix for elements 1 and 2.

(b) Compute the element stiffness matrices for both elements in the global coordinate system.

(c) Assemble the element stiffness matrices and force vectors to the structural matrix equation $[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}$ before applying boundary conditions.

(d) Solve the FE equation after applying the boundary conditions. Write nodal displacements in the global coordinates.

(e) Compute stress in element 1. Is it tensile or compressive?

27. The truss structure shown in the figure supports the force F. The finite element method is used to analyze this structure using two truss elements as shown. The area of cross-section (for all elements) $A = 2\ \text{in}^2$, and Young's modulus $E = 30 \times 10^6$ psi. Both elements are of equal length $L = 10$ ft.

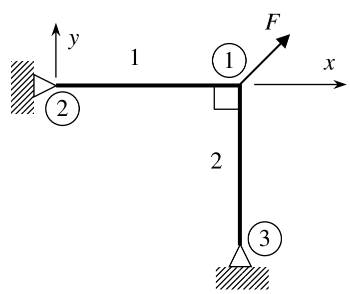
(a) Compute the transformation matrix for elements 1 and 2 between the global coordinate system and the local coordinate system for each element.

(b) Compute the stiffness matrix for the elements 1 and 2.

(c) Assemble the structural matrix equation $[\mathbf{K}_s]\{\mathbf{Q}_s\} = \{\mathbf{F}_s\}$ (without applying the boundary conditions).

(d) It is determined after solving the final equations that the displacement components of node 1 are: $u_{1x} = 1.5 \times 10^{-2}$ in., $u_{1y} = -0.5 \times 10^{-2}$ in. Compute the applied load F.

(e) Compute stress and strain in element 1.



28. In the finite element model of a plane truss in problem 21, the lengths of elements 1 and 2 are 1 m and $\sqrt{2}$ m, respectively. The axial rigidity of the elements is $EA = 10^7$ N.

(a) Fill in the connectivity information of your choice

Element	1st Node i	2nd Node j	φ, deg	l	m
1					
2					

(b) Write down the element stiffness matrix $[k^{(2)}]$ of element 2 with DOFs.

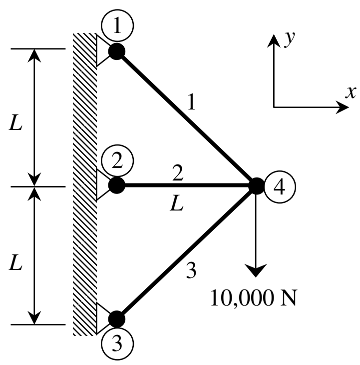
$$\left[k^{(2)}\right] = \begin{bmatrix} \\ \\ \\ \\ \end{bmatrix} \begin{matrix} \square \\ \square \\ \square \\ \square \end{matrix}.$$

(c) Show where $[k^{(2)}]$ will be placed in the structural stiffness matrix $[K_s]$.

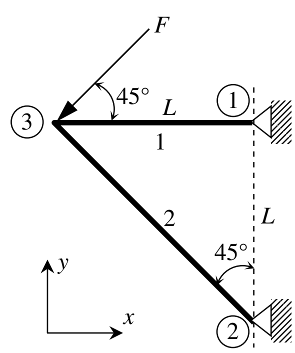
$$[K_s] = \begin{bmatrix} \; & \; & \; & \; & \; & \; \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \\ & & & & & \end{bmatrix} \begin{matrix} u_1 \\ v_1 \\ u_2 \\ v_2 \\ u_2 \\ v_3 \end{matrix}.$$

(d) The displacements are calculated as: $u_1 = +5$ mm, $v_1 = -5\sqrt{2}$ mm. Use your connectivity information to calculate the element force P in element 2.

29. Use the finite element method to solve the plane truss shown below. Assume $AE = 10^6$ N, $L = 1$ m. Determine the nodal displacements, element forces in each element, and the support reactions.



30. The plane truss shown in the figure has two elements and three nodes. Calculate the 4 × 4 element stiffness matrices. Show the row addresses clearly. Derive the final equations (after applying boundary conditions) for the truss in the form of [**K**]{**Q**} = {**F**}. What are nodal displacements and the element forces? Assume: $E = 10^{11}$ Pa, $A = 10^{-4}$ m^2, $L = 1$ m, $F = 14{,}142$ N.

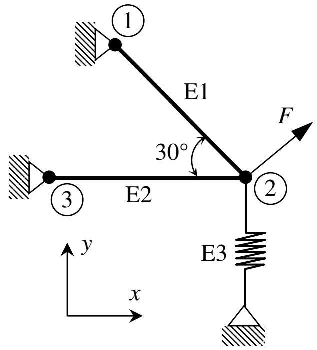


31. Two bars are connected as shown in the figure. Assume all joints are frictionless pin joints. At node 2, a vertical spring is connected as shown. Both bars are of length L and have the same properties: Young's modulus = E and area of cross section = A. The spring stiffness = k.

(a) Set up the stiffness matrices for the two truss elements and the spring element.

(b) Assemble the stiffness matrices to form the global stiffness matrix.

(c) Compute the deflections at node 2, if a force $\mathbf{F} = \{F_x, F_y\}^{\mathrm{T}}$ is applied at node 2.

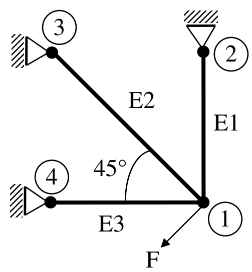


32. The truss structure shown in the figure supports the force F. The finite element method is used to analyze this structure using three truss elements as shown. The area of cross-section (for all elements) $A = 2\ \text{in}^2$. Young's Modulus $E = 30 \times 10^6$ psi. The lengths of the elements are: $L_1 = L_3 = 10$ ft., $L_2 = 14.14$ ft.

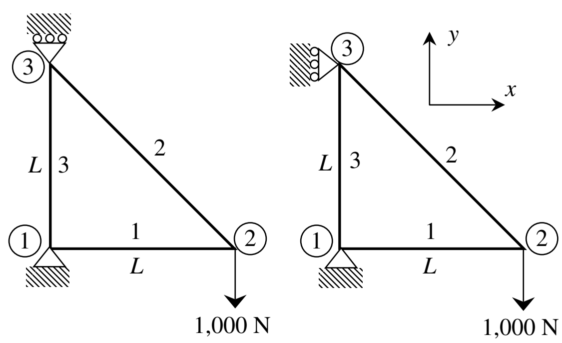
(a) Determine the stiffness matrix of element 2.

(b) It is determined after solving the final equations that the displacement components of node 1 are: $u_1 = -0.5 \times 10^{-2}$ in. and $v_1 = -1.5 \times 10^{-2}$ in. Using the four equations for this element, compute the forces acting on element 2 at nodes 1 and 3. Are these two forces collinear?

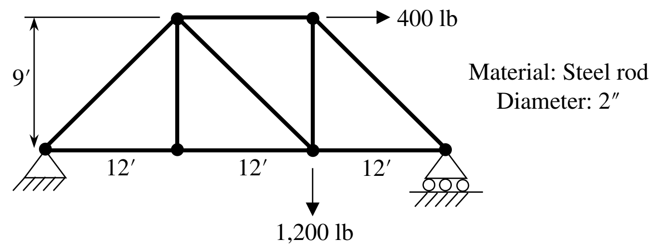
(c) What is the change in length of element 2?



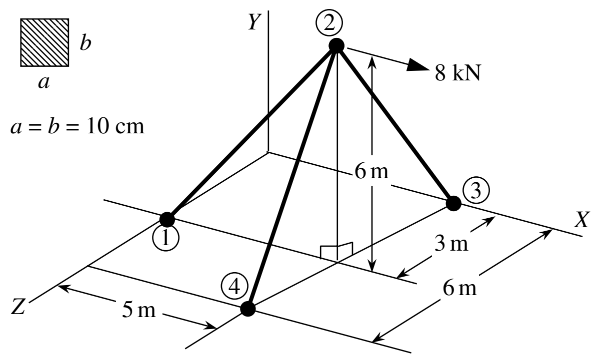
33. It is desired to use the finite element method to solve the two plane truss problems shown in the figure below. Assume $AE = 10^6$ N, $L = 1$ m. Before solving the global equations $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$, find the determinant of $[\mathbf{K}]$. Does $[\mathbf{K}]$ have an inverse? Explain your answer.



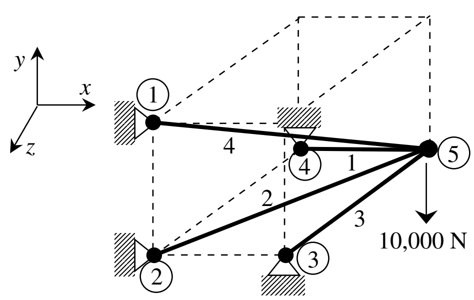
34. Determine the member force and axial stress in each member of the truss shown in the figure using a commercial finite element analysis program. Assume that Young's modulus is 10^4 psi and all cross-sections are circular with a diameter of 2 in. Compare the results with the exact solutions that are obtained from the free-body diagram.



35. Determine the normal stress in each member of the truss structure. All joints are ball-joint, and the material is steel whose Young's modulus is $E = 210$ GPa. Nodes 1, 3, and 4 are pinned on the ground, while node 2 is free to move.



36. The space truss shown has four members. Determine the displacement components of node 5 and the force in each member. The node numbers are numbers in the circle in the figure. The dimensions of the imaginary box that encloses the truss are 1 m × 1 m × 2 m. Assume $AE = 10^6$ N. The coordinates of the nodes are given in the table below:



Node	x	y	z
1	0	1	2
2	0	0	2
3	1	0	2
4	0	0	0
5	1	0	0

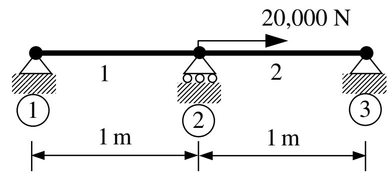
37. The uniaxial bar shown below can be modeled as a one-dimensional bar. The bar has the following properties: $L = 1$ m, $A = 10^{-4}$ m^2, $E = 100$ GPa, and $\alpha = 2 \times 10^{-6}/°C$. From the stress-free initial state, a force of 5,000 N is applied at node 2, and the temperature is lowered by 50 °C below the reference temperature.

(a) Calculate the global matrix equation after applying boundary conditions.

(b) Solve for the displacement u_2.

(c) What is the element force P in the bar?

38. In the structure shown below, the temperature of element 2 is 50 °C above the reference temperature. An external force of 20,000 N is applied in the x direction (horizontal direction) at node 2. Assume $E = 10^{11}$ Pa, $A = 10^{-4}$ m^2, and $\alpha = 2 \times 10^{-6}$/°C.



(a) Write down the stiffness matrices and thermal force vectors for each element.

(b) Write down the global matrix equations.

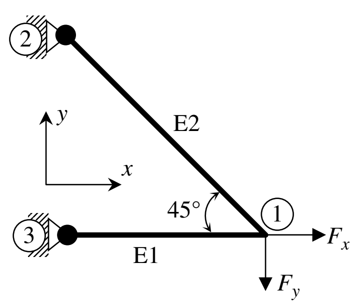
(c) Solve the global equations to determine the displacement at node 2.

(d) Determine the forces in each element. State whether it is tension or compression.

(e) Show that force equilibrium is satisfied at node 2.

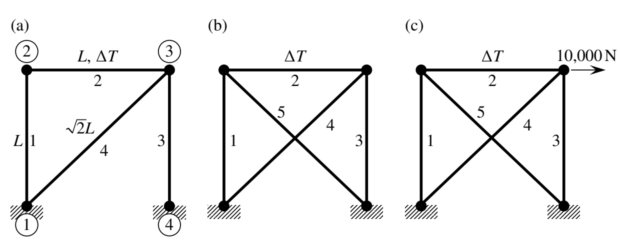
39. The element properties of a plane truss are given in the table below. Derive the global equations in the form: $[\mathbf{K}]_{(2\times2)}\{\mathbf{Q}_{(2\times1)}\} = \{\mathbf{F}_{(2\times1)}\} + \{\mathbf{F}_{T(2\times1)}\}$. Do not solve the equations.

Elem.	$i \to j$	ϕ	l	m	L	A	E	α	ΔT
1					$\sqrt{2}$	1	10	2	10
2					1	1	10	2	−20



40. The three-bar truss problem in figure 1.23 is under a vertical load of $F_y = -5{,}000$ N at node 4 in addition to the temperature change. Solve the problem by (a) applying F_y and ΔT at the same time, and (b) superposing two solutions: one from applying F_y and the other from changing temperature ΔT.

41. Use the finite element method to determine the nodal displacements in the plane truss shown in figure (a). The temperature of element 2 is 200 °C above the reference temperature, that is, $\Delta T^{(2)} = 200$ °C. Compute the force in each element. Show that the force equilibrium is satisfied at node 3. Assume $L = 1$ m, $AE = 10^7$ N, $\alpha = 5 \times 10^{-6}$/°C.

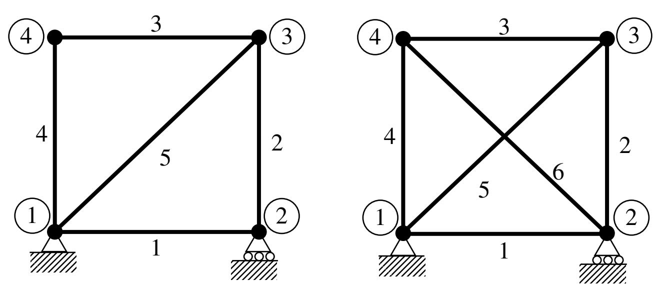


42. Repeat problem 41 for the new configuration with element 5 added, as shown in figure (b)

43. Repeat problem 42 with an external force added to node 3, as shown in figure (c).

44. The properties of the members of the truss in the left side of the figure are given in the table. Calculate the nodal displacement and element forces. Show that force equilibrium is satisfied as node 3.

Elem	L (m)	A (cm^2)	E (GPa)	α (/°C)	ΔT (°C)
1	1	1	100	20×10^{-6}	0
2	1	1	100	20×10^{-6}	0
3	1	1	100	20×10^{-6}	0
4	1	1	100	20×10^{-6}	0
5	$\sqrt{2}$	1	100	20×10^{-6}	–100



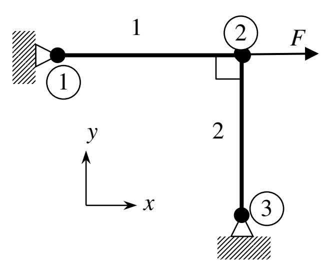
45. Repeat problem 44 for the truss on the right side of the figure. The properties of element 6 are same as those of element 5, but $\Delta T = 0$ °C.

46. The truss shown in the figure supports the force $F = 1{,}000$ N. Both elements have the same axial rigidity of $AE = 10^7$ N, thermal expansion coefficient of $\alpha = 2 \times 10^{-6}$/°C, and length $L = 1$ m. While the temperature at element 1 remains constant, that of element 2 is dropped by 50 °C.

(a) Write the 4 × 4 element stiffness matrices [**k**] and the 4 × 1 thermal force vectors {$\mathbf{f}_T$} for elements 1 and 2. Show the row addresses clearly.

(b) Assemble two elements and apply the boundary conditions to obtain the global matrix equation in the form of [**K**]{**Q**} = {**F**} + {$\mathbf{F}_T$}.

(c) Solve for the nodal displacements.



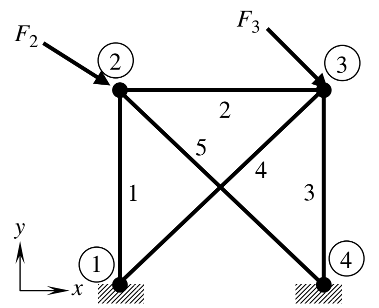
47. The finite element method was used to solve the truss problem shown below. The solution for displacements was obtained as $u_2 = 1.5$ mm, $v_2 = -1.5$ mm, $u_3 = 3$ mm, and $v_3 = -1.5$ mm. Note: the displacements are given in mm.

(a) Determine the axial forces P in elements 2 and 4.

(b) The forces in elements 3 and 5 are found to be as follows: $P^{(3)} = -2{,}000$ N, $P^{(5)} = 7{,}070$ N. Determine the support reactions R_{y4} at node 4 using the nodal equilibrium equations.

The element properties are listed in the following table.

Elem	$i \rightarrow J$	AE [N]	L [m]	ΔT [°C]	α [1/°C]	ϕ
1	1, 2	10^7	1	−100	10^{-6}	90°
2	2, 3	10^7	1	0	10^{-6}	0°
3	3, 4	10^7	1	+100	10^{-6}	−90°
4	1, 3	10^7	1.414	+200	10^{-6}	45°
5	2, 4	10^7	1.414	+200	10^{-6}	−45°



48. Use the finite element method to solve the plane truss shown in problem 29. Assume $AE = 2 \times 10^6$ N, $L = 1$ m, $\alpha = 10 \times 10^{-6}$/°C. The temperature of element 1 is 100 °C below the reference temperature, while elements 2 and 3 are at the reference temperature. Determine the nodal displacements, forces in each element, and the support reactions. Show that the nodal equilibrium is satisfied at node 4.