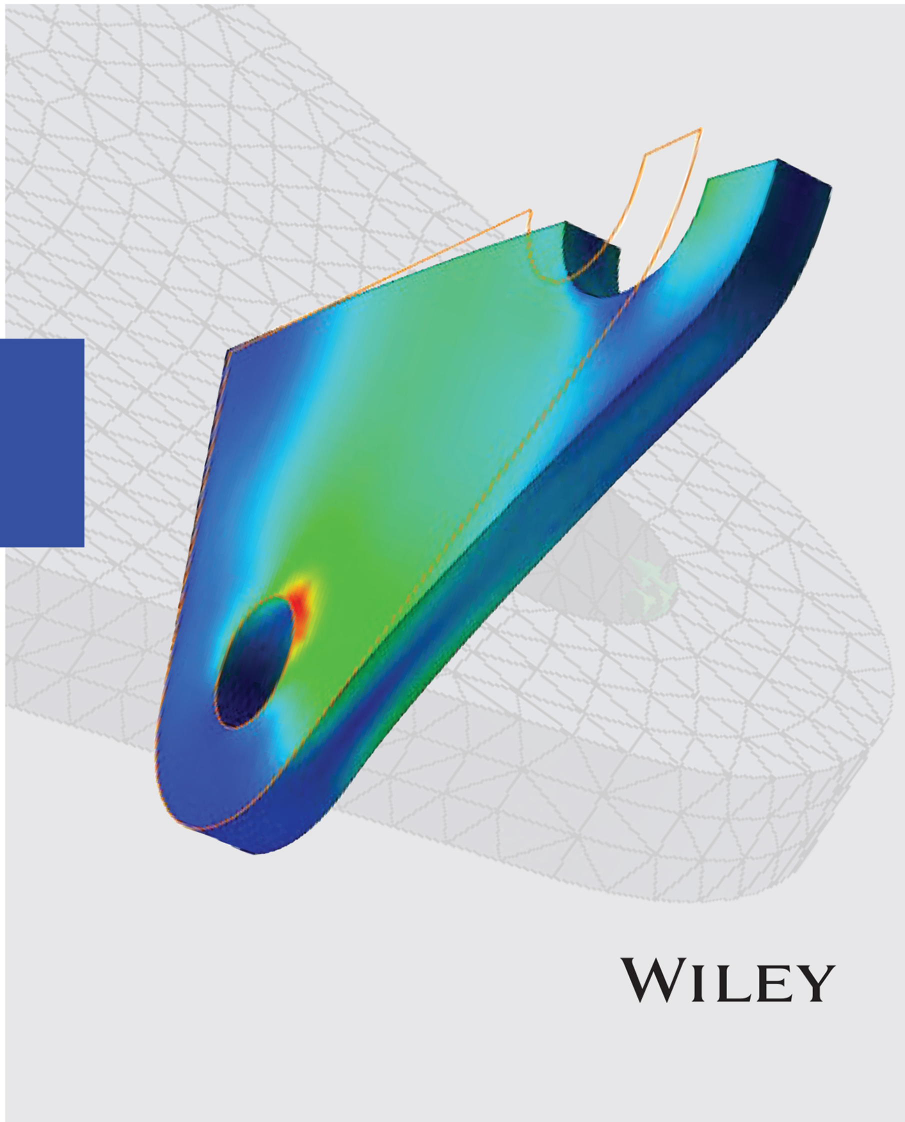
INTRODUCTION TO FINITE ELEMENT ANALYSIS AND DESIGN



NAM H. KIM
BHAVANI V. SANKAR
ASHOK V. KUMAR

2ND EDITION

with website

WILEY

Introduction to Finite Element Analysis and Design

Introduction to Finite Element Analysis and Design

Second Edition

Nam H. Kim, Bhavani V. Sankar, and Ashok V. Kumar
University of Florida

WILEY

This edition first published 2018

Edition History
John Wiley & Sons Ltd (1e, 2008)

Registered Offices
John Wiley & Sons, Inc., 111 River Street, Hoboken, NJ 07030, USA
John Wiley & Sons Ltd, The Atrium, Southern Gate, Chichester, West Sussex, PO19 8SQ, UK

Editorial Office
The Atrium, Southern Gate, Chichester, West Sussex, PO19 8SQ, UK

For details of our global editorial offices, customer services, and more information about Wiley products visit us at www.wiley.com.

Wiley also publishes its books in a variety of electronic formats and by print-on-demand. Some content that appears in standard print versions of this book may not be available in other formats.

Library of Congress Cataloging-in-Publication Data

Names: Kim, Nam H., author. | Sankar, Bhavani V., author. | Kumar, Ashok V., author.
Title: Introduction to finite element analysis and design / by Nam H. Kim, Bhavani V. Sankar, Ashok V. Kumar.
Description: Second edition. | Hoboken, NJ : John Wiley & Sons, 2018. | Includes bibliographical references and index. |
Identifiers: LCCN 2018002473 (print) | LCCN 2018007147 (ebook) | ISBN 9781119078746 (pdf) | ISBN 9781119078739 (epub) | ISBN 9781119078722 (cloth)
Subjects: LCSH: Finite element method. | Engineering mathematics.
Classification: LCC TA347.F5 (ebook) | LCC TA347.F5 K56 2018 (print) | DDC 620.001/51825–dc23
LC record available at https://lccn.loc.gov/2018002473

Cover Design: Wiley
Cover Image: Courtesy of Nam H. Kim

Set in 10/12pt Times by SPi Global, Pondicherry, India

Printed and bound by CPI Group (UK) Ltd, Croydon, CR0 4YY

10 9 8 7 6 5 4 3 2 1

To our wives, Jeehyun, Mira, and Gouri

Contents

Preface

Finite Element Method (FEM) is a numerical method for solving differential equations that describe many engineering problems. 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 a small 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 FEM.

Many universities teach FEM to students at the junior/senior level. One of the biggest challenges to the instructor is finding a textbook appropriate to the level of students. In the past, FEM was taught only to graduate students who would carry out research in that field. Accordingly, many textbooks focus on theoretical development and numerical implementation of the method. However, the goal of an undergraduate FEM course is to introduce the basic concepts so that the students can use the method efficiently and interpret the results properly. Furthermore, the theoretical aspects of FEM must be presented without too much mathematical niceties. Practical applications through several design projects can help students to understand the method clearly.

This book is suitable for junior/senior level undergraduate students and beginning graduate students in engineering mechanics, mechanical, civil, aerospace, biomedical and industrial engineering as well as researchers and design engineers in the above fields.

The textbook is organized into ten chapters. The Appendix at the end summarizes most mathematical preliminaries that are repeatedly used in the text. The Appendix is by no means a comprehensive mathematical treatment of the subject. Rather, it provides a common notation and the minimum amount of mathematical knowledge that will be required in using the book effectively. This includes basics of matrix algebra, minimization of quadratic functions, and techniques for solving linear equations that are commonly used in commercial finite element programs.

The book begins with the introduction of finite element concepts via the direct stiffness method using spring elements. The concepts of nodes, elements, internal forces, equilibrium, assembly, and applying boundary conditions are presented in detail. The spring element is then extended to the uniaxial bar element without introducing interpolation. The concept of local (elemental) and global coordinates and their transformations and element connectivity tables are introduced via two– and three–dimensional truss elements. Four design projects are provided at the end of the chapter, so that students can apply the method to real life problems. The direct method in Chapter 1 provides a clear physical insight into 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 direct stiffness method becomes impractical for more realistic problems especially multi-dimensional problems. In Chapter 2, we introduce more general approaches, such as, the Weighted Residual Methods and, in particular, the Galerkin Method. Similarity to energy methods in solid and structural mechanics problems is discussed. We include a simple 1–D variational formulation in Chapter 2 using boundary value problems. The concept of polynomial approximation and domain discretization is introduced. The formal procedure of finite element analysis is also presented in this chapter. Chapter 2 is written in such way that it can be left out in elementary level courses.

The 1–D formulation is further extended to beams and plane frames in Chapter 3. At this point, the direct method is not useful because the stiffness matrix generated from the direct method cannot provide a clear physical interpretation. Accordingly, we use the principle of minimum potential energy to derive the matrix equation at the element level. The 1–D beam element is extended to 2–D frame element by using coordinate transformation. A 2–D bicycle frame design project is included at the end of this chapter. Buckling of beams and plane frames is included in the revised second edition. First, the concepts of linear buckling of beam is introduced using the Rayleigh-Ritz method. Then the corresponding energy terms are derived in the finite element context.

The finite element formulation is extended to the steady–state heat transfer problem in Chapter 4. Both direct and Galerkin's methods along with convective boundary conditions are included. Two-dimensional heat transfer problems are discussed in the second edition. Practical issues in modeling 2D heat transfer problems are also discussed.

Before proceeding to solid elements in Chapter 6, a review of solid mechanics is provided in Chapter 5. The concepts of stress and strain are presented followed by constitutive relations and equilibrium equations. We limit our interest to linear, isotropic materials in order to make the concepts simple and clear. However, advanced concepts such as transformation of stress and strain, and the eigen value problem for calculating the principal values, are also included. Since, in practice, FEM is used mostly for designing a structure or a mechanical system, failure/yield criteria are also introduced in this chapter.

In Chapter 6, we introduce 2–D solid elements. The governing variational equation is developed using the principle of minimum potential energy. The finite element concepts are explained in detail using only triangular and rectangular elements. Numerical performance of each element is discussed through examples. A new addition to the second edition is the axisymmetric element as it is essentially a plane problem.

The concept of isoparametric mapping is introduce in a separate chapter (Chapter 7) as most practical problems require irregular elements such as linear or higher order quadrilateral elements. Three-dimensional solid elements are introduced in this chapter. Numerical integration and FE modeling practices for isoparametric elements are also included.

Dynamic problems is another addition to the second edition. The concept of free vibration, calculation of natural frequencies and mode shapes, various time integration methods and mode superposition method, are all explained using 1-D structural elements such as uniaxial bars and beams.

In Chapter 9, we discuss traditional finite element analysis procedures, including preliminary analysis, pre-processing, solving matrix equations, and post-processing. Emphasis is on selection of element types, approximating the part geometry, different types of meshing, convergence, and taking advantage of symmetry. A design project involving 2–D analysis is provided at the end of the chapter.

As one of the important goals of FEM is to use the tool for engineering design, the last chapter (Chapter 10) is dedicated to the topic of structural design using FEM. The basic concept of design parameterization and the standard design problem formulation are presented. This chapter is self contained and can be skipped depending on the schedule and content of the course.

Each chapter contains a comprehensive set of homework problems, some of which require commercial FEA programs. A total of nine design projects are provided in the book.

We are thankful to several instructors across the country who used the first edition and provided feedback. We are grateful for their valuable suggestions especially regarding example and exercise problems.

September 2017
Nam H. Kim, Bhavani V. Sankar and Ashok V. Kumar

About the Companion Website

This book is accompanied by a companion website:

www.wiley.com/go/kim/finite_element_analysis_design

The website includes:

- Programs
- Exercise problems

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

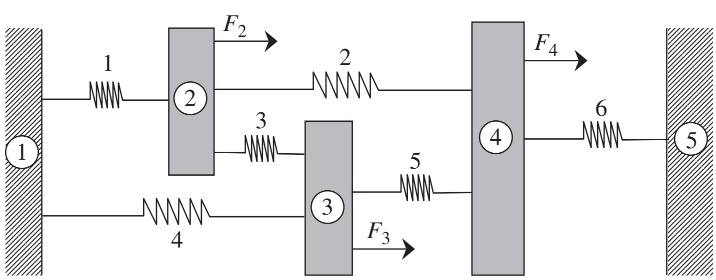


Figure 1.1 Rigid bodies connected by springs

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

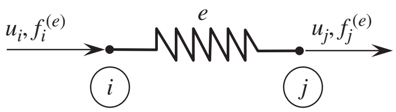


Figure 1.2 Spring element (e) connected by node i and node j

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.

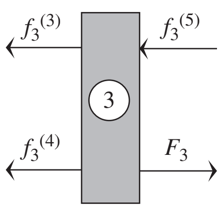


Figure 1.3 Free-body diagram of node 3 in the example shown in figure 1.1. The external force, F_3, and the forces, $f_3^{(e)}$, exerted by the springs attached to the node are shown. Note the forces $f_3^{(e)}$ act in the negative direction.

[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:

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

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.

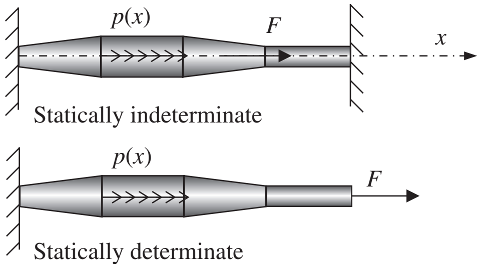


Figure 1.4 Typical one dimensional bar problems

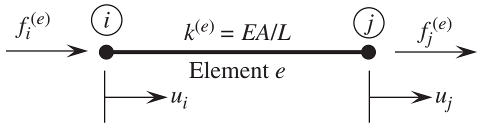


Figure 1.5 Uniaxial bar finite element

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 $u_1, u_2, u_3, \ldots, u_N$, 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

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

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

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

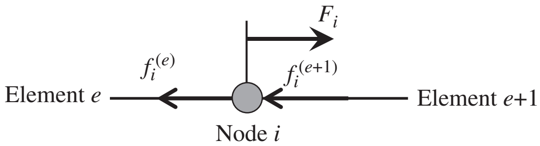


Figure 1.6 Force equilibrium at node i

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

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

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

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

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

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:

$$10^7 \begin{array}{c} \begin{array}{ccc} u_1 & u_2 & u_3 \end{array} \\ \begin{bmatrix} 4 & -4 & 0 \\ -4 & 4+5 & -5 \\ 0 & -5 & 5 \end{bmatrix} \end{array} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 10{,}000 \\ F_3 \end{Bmatrix}. \qquad (1.24)$$

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 $[\mathbf{K}]\{\mathbf{Q}\} = \{\mathbf{F}\}$:

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

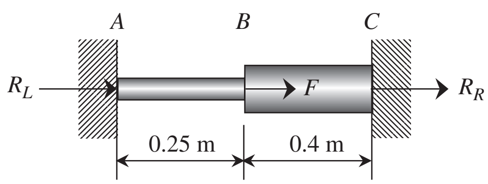


Figure 1.7 Two clamped uniaxial bars

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: $\{\mathbf{Q}_s\}^T = \{u_1, u_2, u_3\} = \{0,\ 1.111 \times 10^{-4},\ 0\}$. 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

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

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

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

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.

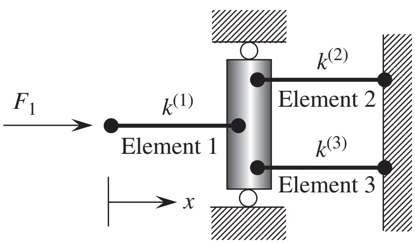


Figure 1.8 One-dimensional structure with three uniaxial bar elements

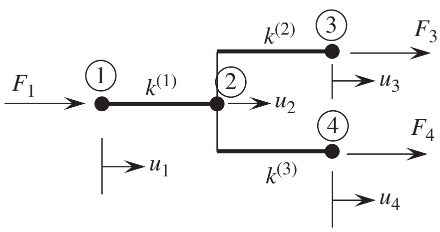


Figure 1.9 Finite element model

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

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

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

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

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

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:

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

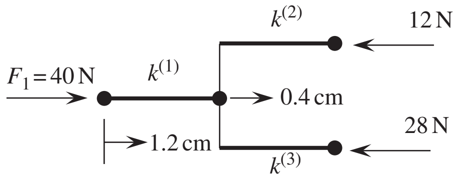


Figure 1.10 Free-body diagram of the structure

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

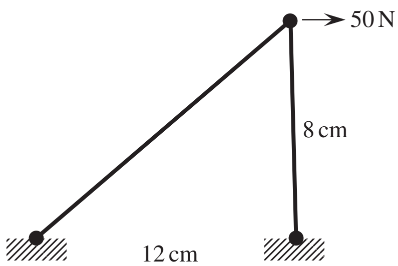


Figure 1.11 A plane truss consisting of two members

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.

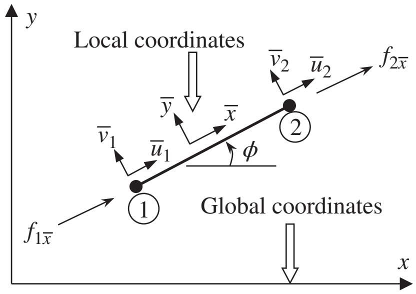


Figure 1.12 Local and global coordinate systems