FINITE ELEMENT ANALYSIS

FINITE ELEMENT ANALYSIS
Theory and Application with ANSYS

Saeed Moaveni
Minnesota State University, Mankato

PRENTICE HALL, Upper Saddle River, New Jersey 07458

Library of Congress Cataloging-in-Publication Data

Moaveni, Saeed.
Finite element analysis. Theory and application with ANSYS
p. cm.
Includes bibliographical references and index.
ISBN 0-13-785098-0
1. Finite element method—Data processing. 2. ANSYS (Computer system) I. Title
TA347.F5M62 1999
620'.001'51535—dc21
98-31163
CIP

Executive editor: *BILL STENQUIST*
Production editor: *CAROLE SURACI*
Editor-in-chief: *MARCIA HORTON*
Managing editor: *EILEEN CLARK*
Marketing manager: *DANNY HOYT*
Assistant vice president of production and manufacturing: *DAVID W. RICCARDI*
Cover designer: *BRUCE KENSELAAR*
Copy editor: *ABIGAIL BAKER*
Manufacturing buyer: *PAT BROWN*
Editorial assistant: *MEG WEIST*

Printed in the United States of America

10 9 8 7 6 5 4 3 2

ISBN 0-13-785098-0

Prentice-Hall International (UK) Limited, *London*
Prentice-Hall of Australia Pty. Limited, *Sydney*
Prentice-Hall Canada, Inc., *Toronto*
Prentice-Hall Hispanoamericana, S.A., *Mexico*
Prentice-Hall of India Private Limited, *New Delhi*
Prentice-Hall of Japan, Inc., *Tokyo*
Simon & Schuster Asia Pte. Ltd., *Singapore*
Editora Prentice-Hall do Brasil, Ltda., *Rio de Janeiro*

To my mother and father

Contents

Preface

There are many good textbooks already in existence that cover the theory of finite element methods for advanced students. However, none of these books incorporate ANSYS as an integral part of their materials to introduce finite element modeling to undergraduate students and the newcomers. In recent years, the use of finite element analysis as a design tool has grown rapidly. Easy to use comprehensive packages such as ANSYS, a general-purpose finite element computer program, have become common tools in the hands of design engineers. Unfortunately, many engineers who lack the proper training or understanding of the underlying concepts have been using these tools. This introductory book is written to assist engineering students and practicing engineers new to the field of finite element modeling to gain a clear understanding of the basic concepts. The text offers insight into the theoretical aspects of finite element analysis and also covers some practical aspects of modeling. Great care has been exercised to avoid overwhelming students with theory. Yet enough theoretical background is offered to allow individuals to use ANSYS intelligently and effectively. ANSYS is an integral part of this text. In each chapter, the relevant basic theory is discussed first and demonstrated using simple problems with hand-calculations. These problems are followed by examples which are solved using ANSYS. Exercises in the text are also presented in this manner. Some exercises require manual calculations while others, more complex in nature, require the use of ANSYS. The simpler hand-calculation problems will enhance students' understanding of the concepts by encouraging them to go through the necessary steps in a finite element analysis. Design problems are also included at the end of Chapters 2, 4, 7, 8, and 10.

Various sources of error that can contribute to wrong results are discussed. A good engineer must always find ways to check the results. While experimental testing of models may be the best way, such testing may be expensive or time consuming. Therefore, whenever possible, throughout this text emphasis is placed on doing a "sanity check" to verify one's Finite Element Analysis (FEA). A section at the end of each appropriate chapter is devoted to possible approaches for verifying ANSYS results.

Another unique feature of this book is that the last chapter is devoted to the introduction of design optimization and parametric programming with ANSYS.

The book is organized into 11 chapters. Chapter 1 reviews basic ideas in finite element analysis. Common formulations, such as direct, potential energy, and weighted residual methods are discussed. Chapter 2 deals with the analysis of trusses because trusses offer economical solutions to many engineering structural problems. An overview of ANSYS program is given in Chapter 2 so that students can begin to use ANSYS right away. Chapter 3 lays the foundation for analysis of one-dimensional problems by introducing one-dimensional linear, quadratic, and cubic elements. Global, local and natural coordinate systems are also discussed in detail in Chapter 3. An introduction to isoparametric formulation and numerical integration by Gauss-Legendre formulae are also presented in Chapter 3. Chapter 4 considers Galerkin formulation of one-dimensional heat transfer and fluid problems. Minimum total potential energy of solid mechanics problems are also discussed in Chapter 4. Two-dimensional linear and higher order elements are introduced in Chapter 5. Gauss-Legendre formulae for two-dimensional integrals are also presented in Chapter 5. In Chapter 6, the essential capabilities and the organization of the ANSYS program are covered. The basic steps in creating and analyzing a model with ANSYS is discussed in detail. Chapter 7 includes the analysis of two-dimensional heat transfer problems. Chapter 8 provides analysis of torsion of noncircular shafts, beams, frames, and plane stress problems. In Chapter 9, two dimensional ideal fluid mechanics problems are analyzed. Direct formulation of the piping network problems and underground seepage flow are also discussed. Chapter 10 provides a discussion of three-dimensional elements and formulations. This chapter also presents basic ideas regarding top-down and bottom-up solid modeling methods. Design optimization ideas and parametric programming are discussed in Chapter 11. Each chapter begins by stating the objectives and concludes by summarizing what the reader should have gained from studying that chapter.

The examples which are solved using ANSYS show in great detail how to use ANSYS to model and analyze a variety of engineering problems. Chapter 6 is also written in such manner that it can be taught right away if the instructor sees the need to start with ANSYS at the beginning of the course.

A brief review of appropriate fundamental principles in solid mechanics, heat transfer, and fluid mechanics is also provided throughout the book. Additionally, when appropriate, the students are warned about becoming too quick to generate finite element models for problems for which there exist simple analytical solutions. Mechanical and thermophysical properties of some common materials used in engineering are given in Appendices A and B.

Finally, I am planning to maintain a website at http://www.prenhall.com/Moaveni for the following purposes: (1) to share any changes in the upcoming versions of

ANSYS; (2) to share additional information on upcoming text revisions including transient analysis of mechanical and thermal problems and other useful topics that you would like to see covered in the next addition. I am also planning to expand the optimization chapter. Examples with error estimation calculations are also planned; (3) to provide additional homework problems and design problems; and (4) to post, at the website, any corrections that are brought to my attention. The website will be accessible to all students.

Saeed Moaveni

Acknowledgments

I would like to express my sincere gratitude to Mr. Raymond Browell of ANSYS, Inc. for providing the photographs for the cover of this book. Descriptions for the cover photographs are given on page 7. I would also like to thank ANSYS, Inc. for giving me permission to adapt material from various ANSYS documents, related to capabilities and the organization of ANSYS. The essential capabilities and organization of ANSYS are covered in Chapters 2, 6, 10, and 11.

As I have mentioned in the Preface, there are many good published books in finite element analysis. When writing this book, several of these books were consulted. They are cited at the end of each appropriate chapter. The reader can benefit from referring to these books and articles.

I would also like to thank Dr. Nancy Mackenzie of Minnesota State University who made valuable editing suggestions during the first draft of the manuscript. I am also grateful to Mr. Bill Stenquist and Ms. Carole Suraci of Prentice Hall for their assistance in the preparation of this book.

CHAPTER 1

Introduction

The finite element method is a numerical procedure that can be used to obtain solutions to a large class of engineering problems involving stress analysis, heat transfer, electromagnetism, and fluid flow. This book was written to help you gain a clear understanding of the fundamental concepts of finite element modeling. Having a clear understanding of the basic concepts will enable you to use a general-purpose finite element software, such as ANSYS, effectively. ANSYS is an integral part of this text. In each chapter, the relevant basic theory behind each respective concept is discussed first. This discussion is followed by examples that are solved using ANSYS. Throughout this text, emphasis is placed on methods by which you may verify your findings from finite element analysis (FEA). In addition, at the end of particular chapters, a section is devoted to the approaches you should consider to verify results generated by using ANSYS.

Some of the exercises provided in this text require manual calculations. The purpose of these exercises is to enhance your understanding of the concepts by encouraging you to go through the necessary steps of finite element analysis. This book is also written in such a way that it can serve as a reference text for readers who may already be design engineers who are beginning to get involved in finite element modeling and need to know the underlying concepts of FEA.

The objective of this chapter is to introduce you to basic concepts in finite element formulation, including direct formulation, the minimum potential energy theorem, and the weighted residual methods. The main topics of Chapter 1 include the following:

1.1 Engineering Problems
1.2 Numerical Methods
1.3 A Brief History of the Finite Element Method and ANSYS
1.4 Basic Steps in the Finite Element Method
1.5 Direct Formulation
1.6 Minimum Total Potential Energy Formulation
1.7 Weighted Residual Formulations
1.8 Verification of Results
1.9 Understanding the Problem

1.1 ENGINEERING PROBLEMS

In general, engineering problems are mathematical models of physical situations. Mathematical models are differential equations with a set of corresponding boundary and initial conditions. The differential equations are derived by applying the fundamental laws and principles of nature to a system or a control volume. These governing equations represent balance of mass, force, or energy. When possible, the exact solution of these equations renders detailed behavior of a system under a given set of conditions. The analytical solutions are composed of two parts: (1) a homogenous part and (2) a particular part. In any given engineering problem, there are two sets of parameters that influence the way in which a system behaves. First, there are those parameters that provide information regarding the *natural behavior* of a given system. These parameters include properties such as modulus of elasticity, thermal conductivity, and viscosity. Table 1.1 summarizes the physical properties that define the natural characteristics of various problems.

On the other hand, there are parameters that produce *disturbances* in a system. These types of parameters are summarized in Table 1.2. Examples of these parameters include external forces, moments, temperature difference across a medium, and pressure difference in fluid flow.

The system characteristics as shown in Table 1.1 dictate the natural behavior of a system, and they always appear in the *homogenous part of the solution* of a governing differential equation. In contrast, the parameters that cause the disturbances appear in the *particular solution*. It is important to understand the role of these parameters in finite element modeling in terms of their respective appearances in stiffness or conductance matrices and load or forcing matrices. The system characteristics will always show up in the stiffness matrix, conductance matrix, or resistance matrix, whereas the disturbance parameters will always appear in the load matrix.

1.2 NUMERICAL METHODS

There are many practical engineering problems for which we cannot obtain exact solutions. This inability to obtain an exact solution may be attributed to either the complex nature of governing differential equations or the difficulties that arise from dealing with the boundary and initial conditions. To deal with such problems, we resort to numerical approximations. In contrast to analytical solutions, which show the exact behavior of a system at any point within the system, numerical solutions approximate exact solutions only at discrete points, called nodes. The first step of any numerical procedure is discretization. This process divides the medium of interest into a number of small subregions and nodes. There are two common classes of numerical methods: (1) *finite difference methods* and (2) *finite element methods*. With finite difference methods, the differential equation is written for each node, and the derivatives are replaced by *difference equations*. This approach results in a set of simultaneous linear equations. Although finite difference methods are easy to understand and employ in simple problems, they become difficult to apply to problems with complex geometries or complex boundary conditions. This situation is also true for problems with nonisotropic material properties.

TABLE 1.1 Physical properties characterizing various engineering systems

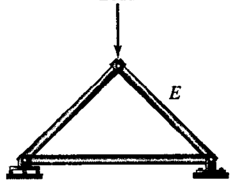
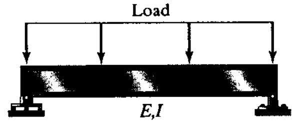
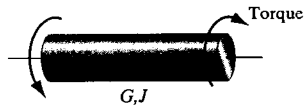
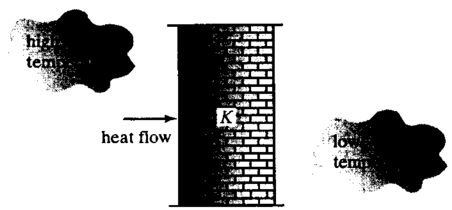
Problem Type	Examples of Parameters That Characterize a System
Solid Mechanics Examples	
Load  a truss	modulus of elasticity, E
 an elastic plate	modulus of elasticity, E
 a beam	modulus of elasticity, E; second moment of area, I
 a shaft	modulus of rigidity, G; polar moment of inertia of the area, J
Heat Transfer Examples	
 a wall	thermal conductivity, K

TABLE 1.1 *(cont.)* Physical properties characterizing various engineering systems

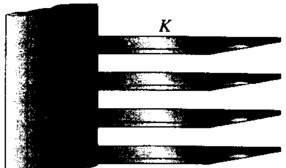
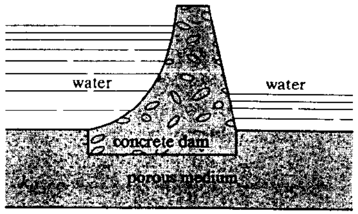
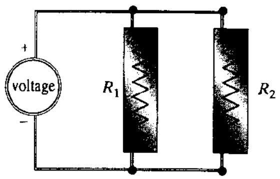
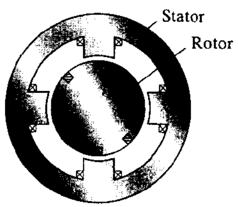
 fins	thermal conductivity, K
Fluid Flow Examples	
 pipe networks	viscosity, μ; relative roughness, e
 a concrete dam	soil permeability, k
Electrical and Magnetism Problems	
 electrical network	resistance, R
 magnetic field of an electric motor	permeability, μ

TABLE 1.2 Parameters causing disturbances in various engineering systems

Problem Type	Examples of Parameters That Produce Disturbances in a System
Solid Mechanics	external forces and moments; support excitation
Heat Transfer	temperature difference; heat input
Fluid Flow and Pipe Networks	pressure difference; rate of flow
Electrical Network	voltage difference

In contrast, the finite element method uses *integral formulations* rather than difference equations to create a system of algebraic equations. Moreover, an approximate continuous function is assumed to represent the solution for each element. The complete solution is then generated by connecting or assembling the individual solutions, allowing for continuity at the interelemental boundaries.

1.3 A BRIEF HISTORY* OF THE FINITE ELEMENT METHOD AND ANSYS

The finite element method is a numerical procedure that can be applied to obtain solutions to a variety of problems in engineering. Steady, transient, linear, or nonlinear problems in stress analysis, heat transfer, fluid flow, and electromagnetism problems may be analyzed with finite element methods. The origin of the modern finite element method may be traced back to the early 1900s, when some investigators approximated and modeled elastic continua using discrete equivalent elastic bars. However, *Courant (1943)* has been credited with being the first person to develop the finite element method. In a paper published in the early 1940s, Courant used piecewise polynomial interpolation over triangular subregions to investigate torsion problems.

The next significant step in the utilization of finite element methods was taken by Boeing in the 1950s when Boeing, followed by others, used triangular stress elements to model airplane wings. Yet, it was not until 1960 that Clough made the term "finite element" popular. During the 1960s, investigators began to apply the finite element method to other areas of engineering, such as heat transfer and seepage flow problems. *Zienkiewicz and Cheung (1967)* wrote the first book entirely devoted to the finite element method in 1967. In 1971, ANSYS was released for the first time.

ANSYS is a comprehensive general-purpose finite element computer program that contains over 100,000 lines of code. ANSYS is capable of performing static, dynamic, heat transfer, fluid flow, and electromagnetism analyses. ANSYS has been a leading FEA program for well over 20 years. The current version of ANSYS has a completely new look, with multiple windows incorporating Graphical User Interface (GUI), pull-down menus, dialog boxes, and a tool bar. Today, you will find ANSYS in use in many engineering fields, including aerospace, automotive, electronics, and nuclear. In order to use ANSYS or any other "canned" FEA computer program intelligently, it is imper-

*See Cook et al. (1989) for more detail.

ative that one first fully understands the underlying basic concepts and limitations of the finite element methods.

ANSYS is a very powerful and impressive engineering tool that may be used to solve a variety of problems. However, a user without a basic understanding of the finite element methods will find himself or herself in the same predicament as a computer technician with access to many impressive instruments and tools, but who cannot fix a computer because he or she does not understand the inner workings of a computer!

1.4 BASIC STEPS IN THE FINITE ELEMENT METHOD

The basic steps involved in any finite element analysis consist of the following:

Preprocessing Phase

1. Create and discretize the solution domain into finite elements; that is, subdivide the problem into nodes and elements.
2. Assume a shape function to represent the physical behavior of an element; that is, an approximate continuous function is assumed to represent the solution of an element.
3. Develop equations for an element.
4. Assemble the elements to present the entire problem. Construct the global stiffness matrix.
5. Apply boundary conditions, initial conditions, and loading.

Solution Phase

6. Solve a set of linear or nonlinear algebraic equations simultaneously to obtain nodal results, such as displacement values at different nodes or temperature values at different nodes in a heat transfer problem.

Postprocessing Phase

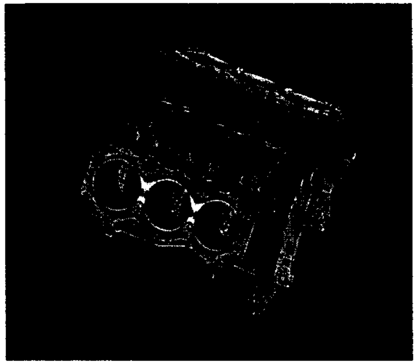
7. Obtain other important information. At this point, you may be interested in values of principal stresses, heat fluxes, etc.

In general, there are several approaches to formulating finite element problems: (1) *Direct Formulation*, (2) *The Minimum Total Potential Energy Formulation*, and (3) *Weighted Residual Formulations*. Again, it is important to note that the basic steps involved in any finite element analysis, regardless of how we generate the finite element model, will be the same as those listed above.

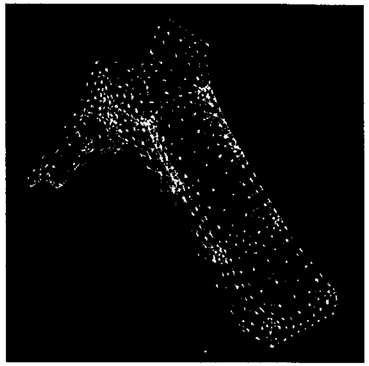
1.5 DIRECT FORMULATION

The following problem illustrates the steps and the procedure involved in direct formulation.

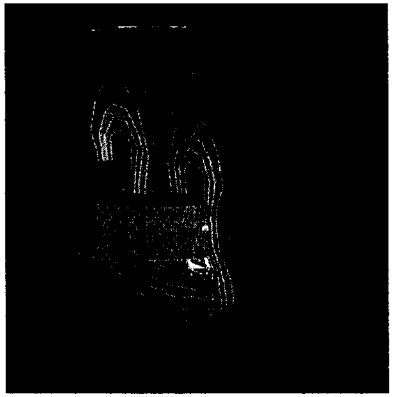
TABLE 1.3 Examples of the capabilities of ANSYS*



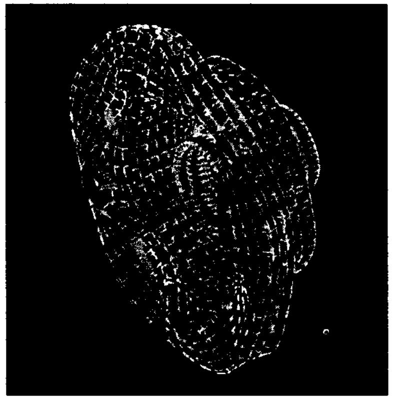
A V6 engine used in front-wheel-drive automobiles was analyzed using ANSYS heat transfer capabilities. The analyses were conducted by Analysis & Design Appl. Co. Ltd. (ADAPCO) on behalf of a major U.S. automobile manufacturer to improve product performance. Contours of thermal stress in the engine block are shown in the figure above.



Large deflection capabilities of ANSYS were utilized by engineers at Today's Kids, a toy manufacturer, to confirm failure locations on the company's play slide, shown in the figure above, when the slide is subjected to overload. This nonlinear analysis capability is required to detect these stresses because of the product's structural behavior.



Electromagnetic capabilities of ANSYS, which include the use of both vector and scalar potentials interfaced through a specialized element, as well as a three-dimensional graphics representation of far-field decay through infinite boundary elements, are depicted in this analysis of a bath plate, shown in the figure above.



Isocontours are used to depict the intensity of the H-field. Structural Analysis Engineering Corporation used ANSYS to determine the natural frequency of a rotor in a disk-brake assembly. In this analysis, 50 modes of vibration, which are considered to contribute to brake sequel, were found to exist in the light-truck brake rotor.

*Photographs courtesy of ANSYS, Inc., Canonsburg, PA

EXAMPLE 1.1

Consider a bar with a variable cross section supporting a load P, as shown in Figure 1.1. The bar is fixed at one end and carries the load P at the other end. Let us designate the width of the bar at the top by w_1, at the bottom by w_2, its thickness by t, and its length by L. The bar's modulus of elasticity will be denoted by E. We are interested in determining how much the bar will deflect at various points along its length when it is subjected to the load P. We will neglect the weight of the bar in the following analysis, assuming that the applied load is considerably larger than the weight of the bar:

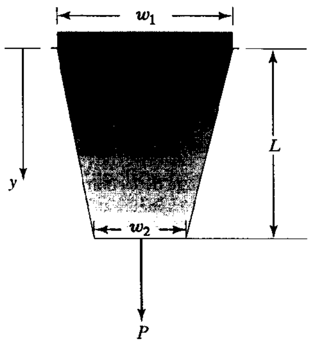


FIGURE 1.1 A bar under axial loading.

Preprocessing Phase

1. *Discretize the solution domain into finite elements.*
We begin by subdividing the problem into nodes and elements. In order to highlight the basic steps in a finite element analysis, we will keep this problem simple and, thus, represent it by a model that has five nodes and four elements, as shown in Figure 1.2. However, note that we can increase the accuracy of our results by generating a model with additional nodes and elements. This task is left as an exercise for you to complete. (See Problem 1 at the end of this chapter.) The given bar is modeled using four individual segments, with each segment having a uniform cross section. The cross-sectional area of each element is represented by an average area of the cross sections at the nodes that make up (define) the element. This model is shown in Figure 1.2.

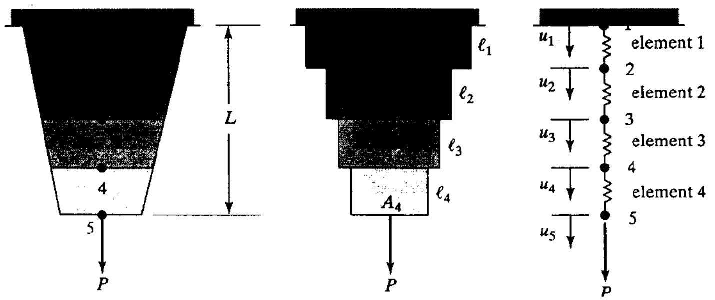


FIGURE 1.2 Subdividing the bar into elements and nodes.

2. *Assume a solution that approximates the behavior of an element.*
In order to study the behavior of a typical element, let's consider the deflection of a solid member with a uniform cross section A that has a length ℓ when subjected to a force F, as shown in Figure 1.3.

The average stress σ in the member is given by

$$\sigma = \frac{F}{A} \tag{1.1}$$

The average normal strain ε of the member is defined as the change in length $\Delta\ell$ per unit original length ℓ of the member:

$$\varepsilon = \frac{\Delta\ell}{\ell} \tag{1.2}$$

Over the elastic region, the stress and strain are related by Hooke's Law, according to the equation

$$\sigma = E\varepsilon \tag{1.3}$$

where E is the modulus of elasticity of the material. Combining Eqs. (1.1), (1.2), and (1.3) and simplifying, we have

$$F = \left(\frac{AE}{\ell}\right)\Delta\ell \tag{1.4}$$

Note that Eq. (1.4) is similar to the equation for a linear spring, $F = kx$. Therefore, a centrally loaded member of uniform cross section may be modeled as a spring with an equivalent stiffness of

$$k_{eq} = \frac{AE}{\ell} \tag{1.5}$$

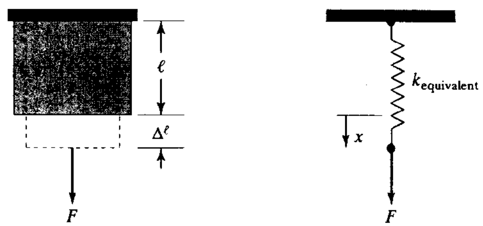


FIGURE 1.3 A solid member of uniform cross section subjected to a force F.

Turning our attention to Example Problem 1.1, we note once again that the bar's cross section varies in the y-direction. As a first approximation, we model the bar as a series of centrally loaded members with different cross sections, as shown in Figure 1.2. Thus, the bar is represented by a model consisting of four elastic springs (elements) in series, and the elastic behavior of an element is modeled by an equivalent linear spring according to the equation

$$f = k_{eq}(u_{i+1} - u_i) = \frac{A_{avg}E}{\ell}(u_{i+1} - u_i) = \frac{(A_{i+1} + A_i)E}{2\ell}(u_{i+1} - u_i) \quad (1.6)$$

where the equivalent element stiffness is given by

$$k_{eq} = \frac{(A_{i+1} + A_i)E}{2\ell} \quad (1.7)$$

A_i and A_{i+1} are the cross-sectional areas of the member at nodes i and $i + 1$, respectively, and ℓ is the length of the element. Employing the above model, let us consider the forces acting on each node. The free body diagram of nodes, which shows the forces acting on nodes 1 through 5 of this model, is depicted in Figure 1.4.

Static equilibrium requires that the sum of the forces acting on each node be zero. This requirement creates the following five equations:

$$\begin{aligned}
&\text{node 1:} \quad R_1 - k_1(u_2 - u_1) = 0 \\
&\text{node 2:} \quad k_1(u_2 - u_1) - k_2(u_3 - u_2) = 0 \\
&\text{node 3:} \quad k_2(u_3 - u_2) - k_3(u_4 - u_3) = 0 \\
&\text{node 4:} \quad k_3(u_4 - u_3) - k_4(u_5 - u_4) = 0 \\
&\text{node 5:} \quad k_4(u_5 - u_4) - P = 0
\end{aligned} \quad (1.8)$$

Rearranging the equilibrium equations given by Eq. (1.8) by separating the reaction force R_1 and the applied external force P from the internal forces, we have:

$$\begin{array}{rrrrrrrrl}
k_1u_1 & -k_1u_2 & & & & & & & = -R_1 \\
-k_1u_1 & +k_1u_2 & +k_2u_2 & -k_2u_3 & & & & & = 0 \\
 & & -k_2u_2 & +k_2u_3 & +k_3u_3 & -k_3u_4 & & & = 0 \\
 & & & & -k_3u_3 & +k_3u_4 & +k_4u_4 & -k_4u_5 & = 0 \\
 & & & & & & -k_4u_4 & +k_4u_5 & = P
\end{array} \quad (1.9)$$

Presenting the equilibrium equations of Eq. (1.9) in a matrix form, we have:

$$\begin{bmatrix} k_1 & -k_1 & 0 & 0 & 0 \\ -k_1 & k_1 + k_2 & -k_2 & 0 & 0 \\ 0 & -k_2 & k_2 + k_3 & -k_3 & 0 \\ 0 & 0 & -k_3 & k_3 + k_4 & -k_4 \\ 0 & 0 & 0 & -k_4 & k_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} -R_1 \\ 0 \\ 0 \\ 0 \\ P \end{Bmatrix} \quad (1.10)$$

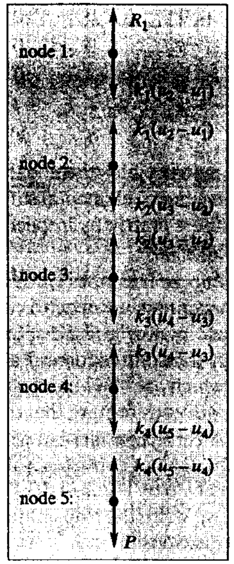


FIGURE 1.4 Free body diagram of the nodes in Example 1.1.

It is also important to distinguish between the reaction forces and the applied loads in the load matrix. Therefore, the matrix relation of Eq. (1.10) can be written as:

$$\begin{Bmatrix} -R_1 \\ 0 \\ 0 \\ 0 \\ 0 \end{Bmatrix} = \begin{bmatrix} k_1 & -k_1 & 0 & 0 & 0 \\ -k_1 & k_1 + k_2 & -k_2 & 0 & 0 \\ 0 & -k_2 & k_2 + k_3 & -k_3 & 0 \\ 0 & 0 & -k_3 & k_3 + k_4 & -k_4 \\ 0 & 0 & 0 & -k_4 & k_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} - \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ P \end{Bmatrix} \quad (1.11)$$

We can readily show that under additional nodal loads and other fixed boundary conditions, the relationship given by Eq. (1.11) can be put into the general form

$$\{\mathbf{R}\} = [\mathbf{K}]\{\mathbf{u}\} - \{\mathbf{F}\} \quad (1.12)$$

which stands for

{reaction matrix} = [stiffness matrix]{displacement matrix} − {load matrix}

Turning our attention to Example 1.1 again, we find that because the bar is fixed at the top, the displacement of node 1 is zero. Thus, the first row of the system of equations given by Eq. (1.10) should read $u_1 = 0$. Thus, application of the boundary condition leads to the following matrix equation:

$$\begin{bmatrix} 1 & 0 & 0 & 0 & 0 \\ -k_1 & k_1 + k_2 & -k_2 & 0 & 0 \\ 0 & -k_2 & k_2 + k_3 & -k_3 & 0 \\ 0 & 0 & -k_3 & k_3 + k_4 & -k_4 \\ 0 & 0 & 0 & -k_4 & k_4 \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ P \end{Bmatrix} \tag{1.13}$$

The solution of the above matrix yields the nodal displacement values. In the next section, we will develop the general elemental stiffness matrix and discuss the construction of the global stiffness matrix by inspection.

3. *Develop equations for an element.*
Because each of the elements in Example 1.1 has two nodes, and with each node we have associated a displacement, we need to create two equations for each element. These equations must involve nodal displacements and the element's stiffness. Consider the internally transmitted forces f_i and f_{i+1} and the end displacements u_i and u_{i+1} of an element, which are shown in Figure 1.5.
Static equilibrium conditions require that the sum of f_i and f_{i+1} be zero. Note that the sum of f_i and f_{i+1} is zero regardless of which representation of Figure 1.5 is selected. However, for the sake of consistency in the forthcoming derivation, we will use the representation given by Figure 1.5(b), so that f_i and f_{i+1} are given in the positive y-direction. Thus, we write the transmitted forces at nodes i and $i + 1$ according to the following equations:

$$f_i = k_{eq}(u_i - u_{i+1}) \tag{1.14}$$
$$f_{i+1} = k_{eq}(u_{i+1} - u_i)$$

Equation (1.14) can be expressed in a matrix form by

$$\begin{Bmatrix} f_i \\ f_{i+1} \end{Bmatrix} = \begin{bmatrix} k_{eq} & -k_{eq} \\ -k_{eq} & k_{eq} \end{bmatrix} \begin{Bmatrix} u_i \\ u_{i+1} \end{Bmatrix} \tag{1.15}$$

4. *Assemble the elements to present the entire problem.*
Applying the elemental description given by Eq. (1.15) to all elements and assembling them (putting them together) will lead to the formation of the global stiffness matrix. The stiffness matrix for element (1) is given by

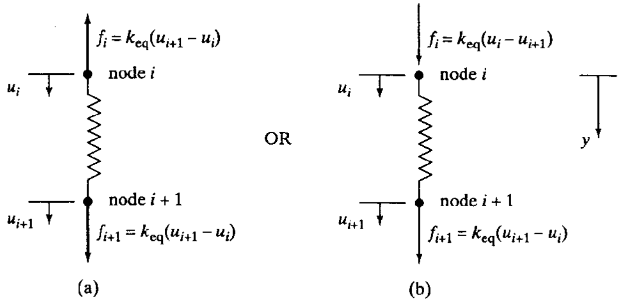


FIGURE 1.5 Internally transmitted forces through an arbitrary element.

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

and its position in the global stiffness matrix is given by

$$[\mathbf{K}]^{(1G)} = \begin{bmatrix} k_1 & -k_1 & 0 & 0 & 0 \\ -k_1 & k_1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{matrix}$$

The nodal displacement matrix is shown alongside the position of element 1 in the global stiffness matrix to aid us to observe the contribution of a node to its neighboring elements. Similarly, for elements (2), (3), and (4), we have

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

and its position in the global matrix

$$[\mathbf{K}]^{(2G)} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & k_2 & -k_2 & 0 & 0 \\ 0 & -k_2 & k_2 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{matrix}$$

$$[\mathbf{K}]^{(3)} = \begin{bmatrix} k_3 & -k_3 \\ -k_3 & k_3 \end{bmatrix}$$

and its position in the global matrix

$$[\mathbf{K}]^{(3G)} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & k_3 & -k_3 & 0 \\ 0 & 0 & -k_3 & k_3 & 0 \\ 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{matrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{matrix}$$

and

$$[\mathbf{K}]^{(4)} = \begin{bmatrix} k_4 & -k_4 \\ -k_4 & k_4 \end{bmatrix}$$

and its position in the global matrix

$$[\mathbf{K}]^{(4G)} = \begin{bmatrix} 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & k_4 & -k_4 \\ 0 & 0 & 0 & -k_4 & k_4 \end{bmatrix} \begin{matrix} u_1 \\ u_2 \\ u_3 \\ u_4 \\ u_5 \end{matrix}$$