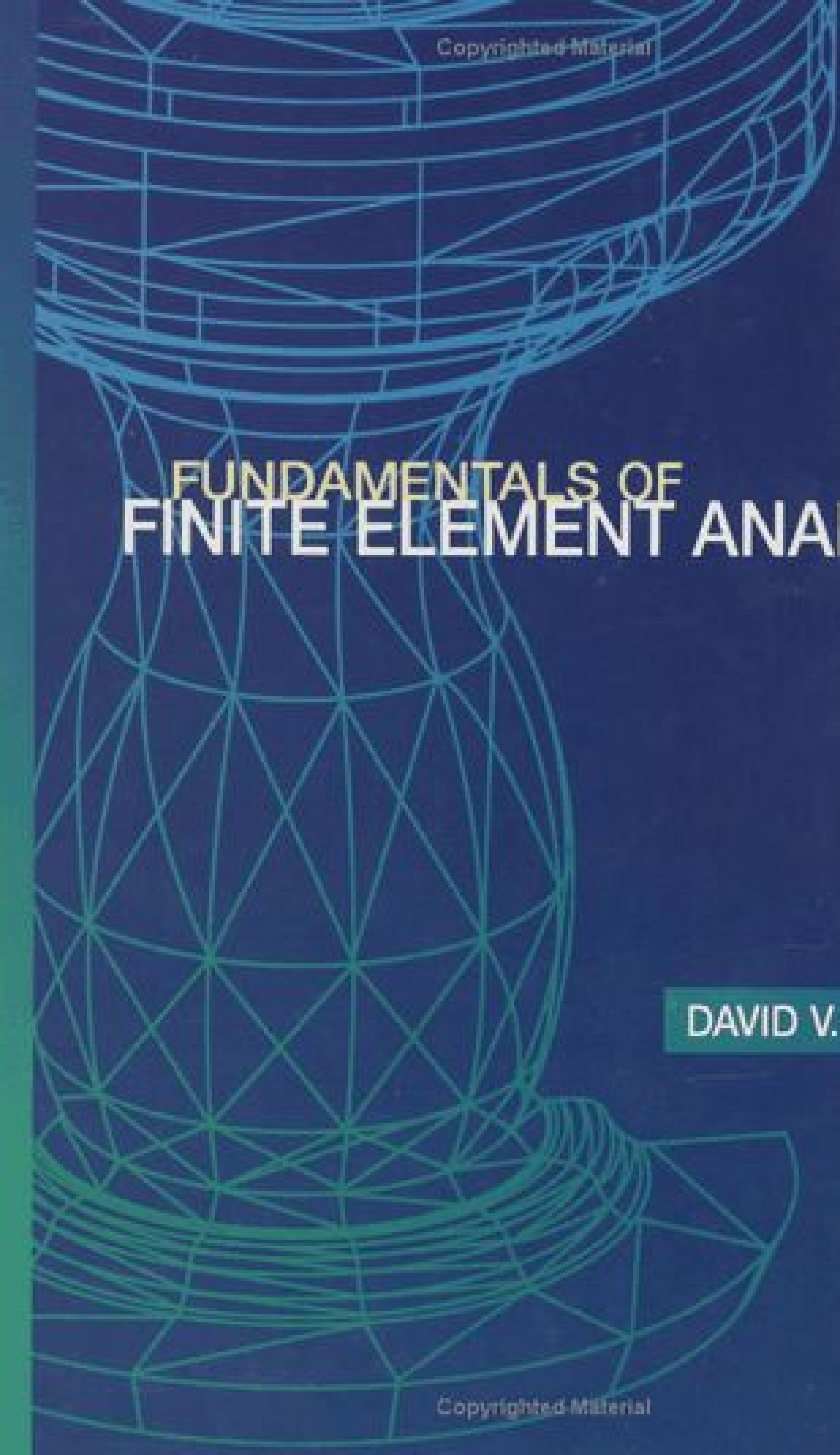


FUNDAMENTALS OF FINITE ELEMENT ANALYSIS

DAVID V. HUTTON

Higher Education

FUNDAMENTALS OF FINITE ELEMENT ANALYSIS

Some ancillaries, including electronic and print components, may not be available to customers outside the United States.

This book is printed on acid-free paper.

International 1 2 3 4 5 6 7 8 9 0 QPF/QPF 0 9 8 7 6 5 4 3
Domestic 1 2 3 4 5 6 7 8 9 0 QPF/QPF 0 9 8 7 6 5 4 3

ISBN 0-07-239536-2
ISBN 0-07-112231-1 (ISE)

Publisher: *Elizabeth A. Jones*
Sponsoring editor: *Jonathan Plant*
Developmental editor: *Lisa Kalner Williams*
Marketing manager: *Sarah Martin*
Senior project manager: *Kay J. Brimeyer*
Production supervisor: *Kara Kudronowicz*
Media project manager: *Jodi K. Banowetz*
Senior media technology producer: *Phillip Meek*
Designer: *K. Wayne Harms*
Cover designer: *Scan Communication Group Inc.*
Senior photo research coordinator: *Lori Hancock*
Compositor: *Interactive Composition Corporation*
Typeface: *10.5/12 Times Roman*
Printer: *Quebecor World Fairfield, PA*

Library of Congress Cataloging-in-Publication Data

Hutton, David V.
Fundamentals of finite element analysis / David V. Hutton. — 1st ed.
p. cm.
ISBN 0-07-239536-2
1. Finite element method. I. Title.

TA347.F5H88 2004
620'.001'51535—dc21 2003048735
CIP

INTERNATIONAL EDITION ISBN 0-07-112231-1

www.mhhe.com

BRIEF TABLE OF CONTENTS

EXPANDED TABLE OF CONTENTS

Appendix C
Solution Techniques for Linear Algebraic Equations 463

Appendix D
The Finite Element Personal Computer Program 473

Appendix E
Problems for Computer Solution 476

Index 488

PREFACE

Fundamentals of Finite Element Analysis is intended to be the text for a senior-level finite element course in engineering programs. The most appropriate major programs are civil engineering, engineering mechanics, and mechanical engineering. The finite element method is such a widely used analysis-and-design technique that it is essential that undergraduate engineering students have a basic knowledge of the theory and applications of the technique. Toward that objective, I developed and taught an undergraduate "special topics" course on the finite element method at Washington State University in the summer of 1992. The course was composed of approximately two-thirds theory and one-third use of commercial software in solving finite element problems. Since that time, the course has become a regularly offered technical elective in the mechanical engineering program and is generally in high demand. During the developmental process for the course, I was never satisfied with any text that was used, and we tried many. I found the available texts to be at one extreme or the other; namely, essentially no theory and all software application, or all theory and no software application. The former approach, in my opinion, represents training in using computer programs, while the latter represents graduate-level study. I have written this text to seek a middle ground.

Pedagogically, I believe that training undergraduate engineering students to use a particular software package without providing knowledge of the underlying theory is a disservice to the student and can be dangerous for their future employers. While I am acutely aware that most engineering programs have a specific finite element software package available for student use, I do not believe that the text the students use should be tied only to that software. Therefore, I have written this text to be software-independent. I emphasize the basic theory of the finite element method, in a context that can be understood by undergraduate engineering students, and leave the software-specific portions to the instructor.

As the text is intended for an undergraduate course, the prerequisites required are statics, dynamics, mechanics of materials, and calculus through ordinary differential equations. Of necessity, partial differential equations are introduced but in a manner that should be understood based on the stated prerequisites. Applications of the finite element method to heat transfer and fluid mechanics are included, but the necessary derivations are such that previous coursework in those topics is not required. Many students will have taken heat transfer and fluid mechanics courses, and the instructor can expand the topics based on the students' background.

Chapter 1 is a general introduction to the finite element method and includes a description of the basic concept of dividing a domain into finite-size subdomains. The finite difference method is introduced for comparison to the finite element method. A general procedure in the sequence of model definition, solution, and interpretation of results is discussed and related to the generally accepted terms of preprocessing, solution, and postprocessing. A brief history of the finite element method is included, as are a few examples illustrating application of the method.

Chapter 2 introduces the concept of a finite element stiffness matrix and associated displacement equation, in terms of interpolation functions, using the linear spring as a finite element. The linear spring is known to most undergraduate students so the mechanics should not be new. However, representation of the spring as a finite element *is* new but provides a simple, concise example of the finite element method. The premise of spring element formulation is extended to the bar element, and energy methods are introduced. The first theorem of Castigliano is applied, as is the principle of minimum potential energy. Castigliano's theorem is a simple method to introduce the undergraduate student to minimum principles without use of variational calculus.

Chapter 3 uses the bar element of Chapter 2 to illustrate assembly of global equilibrium equations for a structure composed of many finite elements. Transformation from element coordinates to global coordinates is developed and illustrated with both two- and three-dimensional examples. The direct stiffness method is utilized and two methods for global matrix assembly are presented. Application of boundary conditions and solution of the resultant constraint equations is discussed. Use of the basic displacement solution to obtain element strain and stress is shown as a postprocessing operation.

Chapter 4 introduces the beam/flexure element as a bridge to continuity requirements for higher-order elements. Slope continuity is introduced and this requires an adjustment to the assumed interpolation functions to insure continuity. Nodal load vectors are discussed in the context of discrete and distributed loads, using the method of work equivalence.

Chapters 2, 3, and 4 introduce the basic procedures of finite-element modeling in the context of simple structural elements that should be well-known to the student from the prerequisite mechanics of materials course. Thus the emphasis in the early part of the course in which the text is used can be on the finite element method without introduction of new physical concepts. The bar and beam elements can be used to give the student practical truss and frame problems for solution using available finite element software. If the instructor is so inclined, the bar and beam elements (in the two-dimensional context) also provide a relatively simple framework for student development of finite element software using basic programming languages.

Chapter 5 is the springboard to more advanced concepts of finite element analysis. The method of weighted residuals is introduced as the fundamental technique used in the remainder of the text. The Galerkin method is utilized exclusively since I have found this method is both understandable for undergraduate students and is amenable to a wide range of engineering problems. The material in this chapter repeats the bar and beam developments and extends the finite element concept to one-dimensional heat transfer. Application to the bar and beam elements illustrates that the method is in agreement with the basic mechanics approach of Chapters 2–4. Introduction of heat transfer exposes the student to additional applications of the finite element method that are, most likely, new to the student.

Chapter 6 is a stand-alone description of the requirements of interpolation functions used in developing finite element models for *any* physical problem. Continuity and completeness requirements are delineated. Natural (serendipity) coordinates, triangular coordinates, and volume coordinates are defined and used to develop interpolation functions for several element types in two- and three-dimensions. The concept of isoparametric mapping is introduced in the context of the plane quadrilateral element. As a precursor to following chapters, numerical integration using Gaussian quadrature is covered and several examples included. The use of two-dimensional elements to model three-dimensional axisymmetric problems is included.

Chapter 7 uses Galerkin's finite element method to develop the finite element equations for several commonly encountered situations in heat transfer. One-, two- and three-dimensional formulations are discussed for conduction and convection. Radiation is not included, as that phenomenon introduces a nonlinearity that undergraduate students are not prepared to deal with at the intended level of the text. Heat transfer with mass transport is included. The finite difference method in conjunction with the finite element method is utilized to present methods of solving time-dependent heat transfer problems.

Chapter 8 introduces finite element applications to fluid mechanics. The general equations governing fluid flow are so complex and nonlinear that the topic is introduced via ideal flow. The stream function and velocity potential function are illustrated and the applicable restrictions noted. Example problems are included that note the analogy with heat transfer and use heat transfer finite element solutions to solve ideal flow problems. A brief discussion of viscous flow shows the nonlinearities that arise when nonideal flows are considered.

Chapter 9 applies the finite element method to problems in solid mechanics with the proviso that the material response is linearly elastic and small deflection. Both plane stress and plane strain are defined and the finite element formulations developed for each case. General three-dimensional states of stress and axisymmetric stress are included. A model for torsion of noncircular sections is developed using the Prandtl stress function. The purpose of the torsion section is to make the student aware that all torsionally loaded objects are not circular and the analysis methods must be adjusted to suit geometry.

Chapter 10 introduces the concept of dynamic motion of structures. It is not presumed that the student has taken a course in mechanical vibrations; as a result, this chapter includes a primer on basic vibration theory. Most of this material is drawn from my previously published text *Applied Mechanical Vibrations*. The concept of the mass or inertia matrix is developed by examples of simple spring-mass systems and then extended to continuous bodies. Both lumped and consistent mass matrices are defined and used in examples. Modal analysis is the basic method espoused for dynamic response; hence, a considerable amount of text material is devoted to determination of natural modes, orthogonality, and modal superposition. Combination of finite difference and finite element methods for solving transient dynamic structural problems is included.

The appendices are included in order to provide the student with material that might be new or may be "rusty" in the student's mind.

Appendix A is a review of matrix algebra and should be known to the student from a course in linear algebra.

Appendix B states the general three-dimensional constitutive relations for a homogeneous, isotropic, elastic material. I have found over the years that undergraduate engineering students do not have a firm grasp of these relations. In general, the student has been exposed to so many special cases that the three-dimensional equations are not truly understood.

Appendix C covers three methods for solving linear algebraic equations. Some students may use this material as an outline for programming solution methods. I include the appendix only so the reader is aware of the algorithms underlying the software he/she will use in solving finite element problems.

Appendix D describes the basic computational capabilities of the FEPC software. The FEPC (FEPfinite element program for the PCpersonal computer) was developed by the late Dr. Charles Knight of Virginia Polytechnic Institute and State University and is used in conjunction with this text with permission of his estate. Dr. Knight's programs allow analysis of two-dimensional programs using bar, beam, and plane stress elements. The appendix describes in general terms the capabilities and limitations of the software. The FEPC program is available to the student at www.mhhe.com/hutton.

Appendix E includes problems for several chapters of the text that should be solved via commercial finite element software. Whether the instructor has available ANSYS, ALGOR, COSMOS, etc., these problems are oriented to systems having many degrees of freedom and not amenable to hand calculation. Additional problems of this sort will be added to the website on a continuing basis.

The textbook features a Web site (www.mhhe.com/hutton) with finite element analysis links and the FEPC program. At this site, instructors will have access to PowerPoint images and an Instructors' Solutions Manual. Instructors can access these tools by contacting their local McGraw-Hill sales representative for password information.

I thank Raghu Agarwal, Rong Y. Chen, Nels Madsen, Robert L. Rankin, Joseph J. Rencis, Stephen R. Swanson, and Lonny L. Thompson, who reviewed some or all of the manuscript and provided constructive suggestions and criticisms that have helped improve the book.

I am grateful to all the staff at McGraw-Hill who have labored to make this project a reality. I especially acknowledge the patient encouragement and professionalism of Jonathan Plant, Senior Editor, Lisa Kalner Williams, Developmental Editor, and Kay Brimeyer, Senior Project Manager.

David V. Hutton

Pullman, WA

CHAPTER 1

Basic Concepts of the Finite Element Method

1.1 INTRODUCTION

The finite element method (FEM), sometimes referred to as *finite element analysis* (FEA), is a computational technique used to obtain approximate solutions of boundary value problems in engineering. Simply stated, a boundary value problem is a mathematical problem in which one or more dependent variables must satisfy a differential equation everywhere within a known domain of independent variables and satisfy specific conditions on the boundary of the domain. Boundary value problems are also sometimes called *field* problems. The field is the domain of interest and most often represents a physical structure. The *field variables* are the dependent variables of interest governed by the differential equation. The *boundary conditions* are the specified values of the field variables (or related variables such as derivatives) on the boundaries of the field. Depending on the type of physical problem being analyzed, the field variables may include physical displacement, temperature, heat flux, and fluid velocity to name only a few.

1.2 HOW DOES THE FINITE ELEMENT METHOD WORK?

The general techniques and terminology of finite element analysis will be introduced with reference to Figure 1.1. The figure depicts a volume of some material or materials having known physical properties. The volume represents the domain of a boundary value problem to be solved. For simplicity, at this point, we assume a two-dimensional case with a single field variable $\phi(x, y)$ to be determined at every point $P(x, y)$ such that a known governing equation (or equations) is satisfied exactly at every such point. Note that this implies an exact

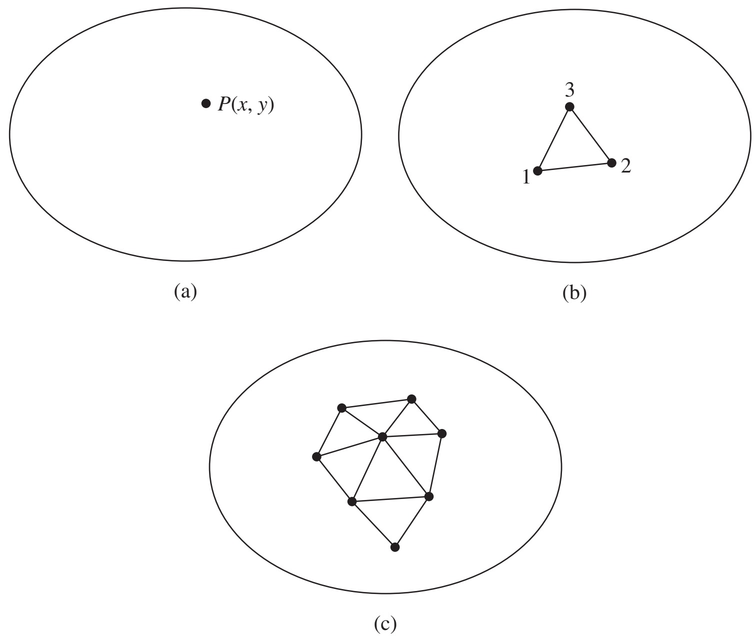


Figure 1.1
(a) A general two-dimensional domain of field variable $\phi(x, y)$.
(b) A three-node finite element defined in the domain. (c) Additional elements showing a partial finite element mesh of the domain.

mathematical solution is obtained; that is, the solution is a closed-form algebraic expression of the independent variables. In practical problems, the domain may be geometrically complex as is, often, the governing equation and the likelihood of obtaining an exact closed-form solution is very low. Therefore, approximate solutions based on numerical techniques and digital computation are most often obtained in engineering analyses of complex problems. Finite element analysis is a powerful technique for obtaining such approximate solutions with good accuracy.

A small triangular element that encloses a finite-sized subdomain of the area of interest is shown in Figure 1.1b. That this element is *not* a differential element of size $dx \times dy$ makes this a *finite element*. As we treat this example as a two-dimensional problem, it is assumed that the thickness in the z direction is constant and z dependency is not indicated in the differential equation. The vertices of the triangular element are numbered to indicate that these points are nodes. A *node* is a specific point in the finite element at which the value of the field variable is to be explicitly calculated. *Exterior* nodes are located on the boundaries of the finite element and may be used to connect an element to adjacent finite elements. Nodes that do not lie on element boundaries are *interior* nodes and cannot be connected to any other element. The triangular element of Figure 1.1b has only exterior nodes.

If the values of the field variable are computed only at nodes, how are values obtained at other points within a finite element? The answer contains the crux of the finite element method: The values of the field variable computed at the nodes are used to approximate the values at nonnodal points (that is, in the element interior) by *interpolation* of the nodal values. For the three-node triangle example, the nodes are all exterior and, at any other point within the element, the field variable is described by the approximate relation

$$\phi(x, y) = N_1(x, y)\phi_1 + N_2(x, y)\phi_2 + N_3(x, y)\phi_3 \tag{1.1}$$

where ϕ_1, ϕ_2, and ϕ_3 are the values of the field variable at the nodes, and N_1, N_2, and N_3 are the *interpolation functions,* also known as *shape functions* or *blending functions*. In the finite element approach, the nodal values of the field variable are treated as unknown *constants* that are to be determined. The interpolation functions are most often polynomial forms of the independent variables, derived to satisfy certain required conditions at the nodes. These conditions are discussed in detail in subsequent chapters. The major point to be made here is that the interpolation functions are predetermined, *known* functions of the independent variables; and these functions describe the variation of the field variable within the finite element.

The triangular element described by Equation 1.1 is said to have 3 *degrees of freedom,* as three nodal values of the field variable are required to describe the field variable everywhere in the element. This would be the case if the field variable represents a scalar field, such as temperature in a heat transfer problem (Chapter 7). If the domain of Figure 1.1 represents a thin, solid body subjected to plane stress (Chapter 9), the field variable becomes the displacement vector and the values of two components must be computed at each node. In the latter case, the three-node triangular element has 6 degrees of freedom. In general, the number of degrees of freedom associated with a finite element is equal to the product of the number of nodes and the number of values of the field variable (and possibly its derivatives) that must be computed at each node.

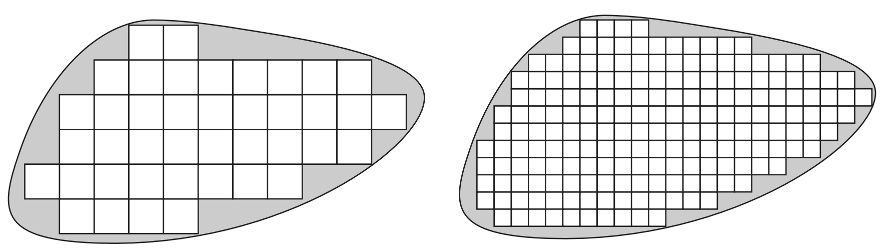
How does this element-based approach work over the entire domain of interest? As depicted in Figure 1.1c, every element is connected *at its exterior nodes* to other elements. The finite element equations are formulated such that, at the nodal connections, the value of the field variable at any connection is the same for each element connected to the node. Thus, continuity of the field variable at the nodes is ensured. In fact, finite element formulations are such that continuity of the field variable across interelement boundaries is also ensured. This feature avoids the physically unacceptable possibility of gaps or voids occurring in the domain. In structural problems, such gaps would represent physical separation of the material. In heat transfer, a “gap” would manifest itself in the form of different temperatures at the same physical point.

Although continuity of the field variable from element to element is inherent to the finite element formulation, interelement continuity of gradients (i.e., derivatives) of the field variable does not generally exist. This is a critical observation. In most cases, such derivatives are of more interest than are field variable values. For example, in structural problems, the field variable is displacement but the true interest is more often in strain and stress. As *strain* is defined in terms of first derivatives of displacement components, strain is not continuous across element boundaries. However, the magnitudes of discontinuities of derivatives can be used to assess solution accuracy and convergence as the number of elements is increased, as is illustrated by the following example.

1.2.1 Comparison of Finite Element and Exact Solutions

The process of representing a physical domain with finite elements is referred to as *meshing,* and the resulting set of elements is known as the finite element *mesh*. As most of the commonly used element geometries have straight sides, it is generally impossible to include the entire physical domain in the element mesh if the domain includes curved boundaries. Such a situation is shown in Figure 1.2a, where a curved-boundary domain is meshed (quite coarsely) using square elements. A refined mesh for the same domain is shown in Figure 1.2b, using smaller, more numerous elements of the same type. Note that the refined mesh includes significantly more of the physical domain in the finite element representation and the curved boundaries are more closely approximated. (Triangular elements could approximate the boundaries even better.)

If the interpolation functions satisfy certain mathematical requirements (Chapter 6), a finite element solution for a particular problem converges to the exact solution of the problem. That is, as the number of elements is increased and the physical dimensions of the elements are decreased, the finite element solution changes incrementally. The incremental changes decrease with the mesh refinement process and approach the exact solution asymptotically. To illustrate convergence, we consider a relatively simple problem that has a known solution. Figure 1.3a depicts a tapered, solid cylinder fixed at one end and subjected to a tensile load at the other end. Assuming the displacement at the point of load application to be of interest, a first approximation is obtained by considering the cylinder to be uniform, having a cross-sectional area equal to the average area



(a) (b)

Figure 1.2
(a) Arbitrary curved-boundary domain modeled using square elements. Stippled areas are not included in the model. A total of 41 elements is shown. (b) Refined finite element mesh showing reduction of the area not included in the model. A total of 192 elements is shown.

of the cylinder (Figure 1.3b). The uniform bar is a *link* or *bar* finite element (Chapter 2), so our first approximation is a one-element, finite element model. The solution is obtained using the strength of materials theory. Next, we model the tapered cylinder as two uniform bars in series, as in Figure 1.3c. In the two-element model, each element is of length equal to half the total length of the cylinder and has a cross-sectional area equal to the average area of the corresponding half-length of the cylinder. The mesh refinement is continued using a four-element model, as in Figure 1.3d, and so on. For this simple problem, the displacement of the end of the cylinder for each of the finite element models is as shown in Figure 1.4a, where the dashed line represents the known solution. Convergence of the finite element solutions to the exact solution is clearly indicated.

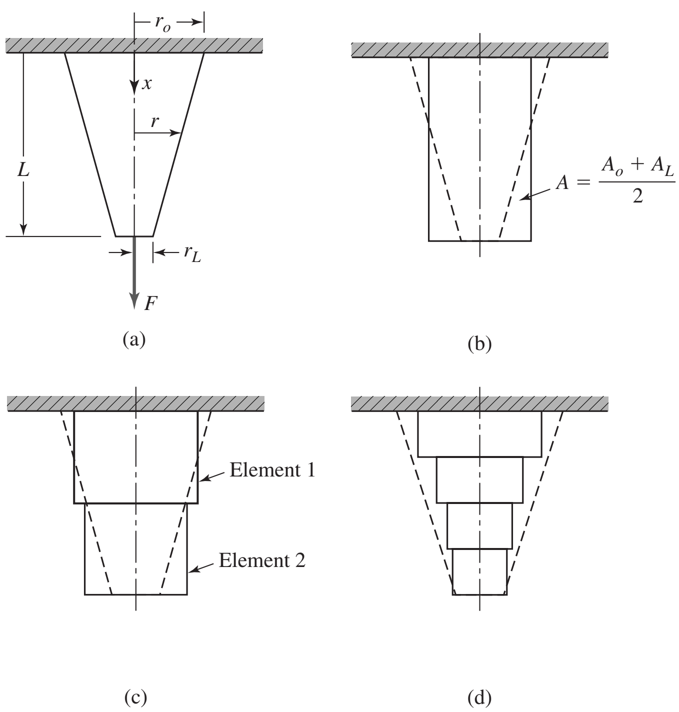


Figure 1.3
(a) Tapered circular cylinder subjected to tensile loading: $r(x) = r_0 - (x/L)(r_0 - r_L)$. (b) Tapered cylinder as a single axial (bar) element using an average area. Actual tapered cylinder is shown as dashed lines. (c) Tapered cylinder modeled as two, equal-length, finite elements. The area of each element is average over the respective tapered cylinder length. (d) Tapered circular cylinder modeled as four, equal-length finite elements. The areas are average over the respective length of cylinder (element length = $L/4$).

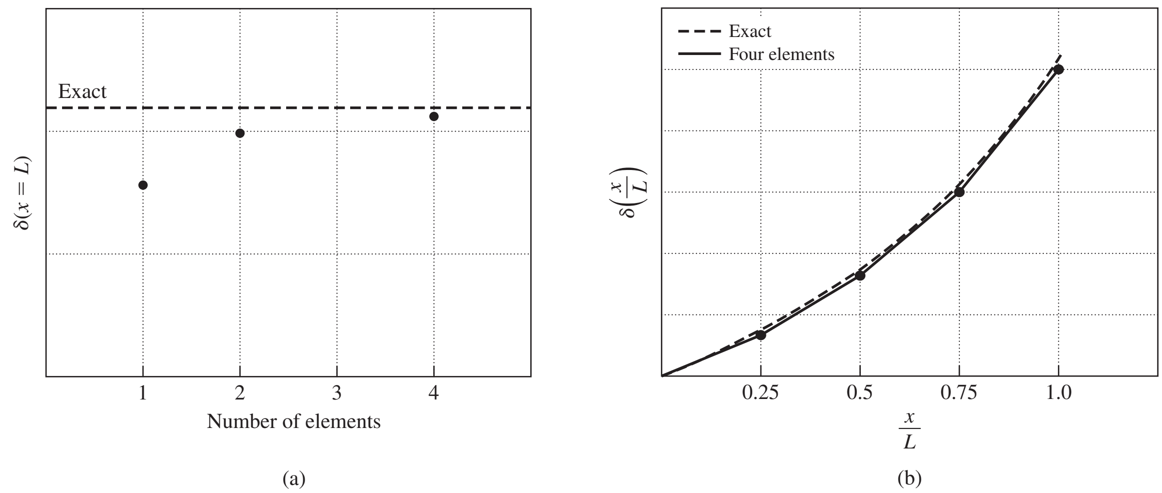


Figure 1.4
(a) Displacement at $x = L$ for tapered cylinder in tension of Figure 1.3. (b) Comparison of the exact solution and the four-element solution for a tapered cylinder in tension.

On the other hand, if we plot displacement as a function of position along the length of the cylinder, we can observe convergence as well as the approximate nature of the finite element solutions. Figure 1.4b depicts the exact strength of materials solution and the displacement solution for the four-element models. We note that the displacement variation in each element is a linear approximation to the true nonlinear solution. The linear variation is directly attributable to the fact that the interpolation functions for a bar element are linear. Second, we note that, as the mesh is refined, the displacement solution converges to the nonlinear solution at *every point* in the solution domain.

The previous paragraph discussed convergence of the displacement of the tapered cylinder. As will be seen in Chapter 2, displacement is the primary field variable in structural problems. In most structural problems, however, we are interested primarily in stresses induced by specified loadings. The stresses must be computed via the appropriate stress-strain relations, and the strain components are derived from the displacement field solution. Hence, strains and stresses are referred to as *derived* variables. For example, if we plot the element stresses for the tapered cylinder example just cited for the exact solution as well as the finite element solutions for two- and four-element models as depicted in Figure 1.5, we observe that the stresses are constant in each element and represent a *discontinuous* solution of the problem in terms of stresses and strains. We also note that, as the number of elements increases, the jump discontinuities in stress decrease in magnitude. This phenomenon is characteristic of the finite element method. The formulation of the finite element method for a given problem is such that the primary field variable is continuous from element to element but

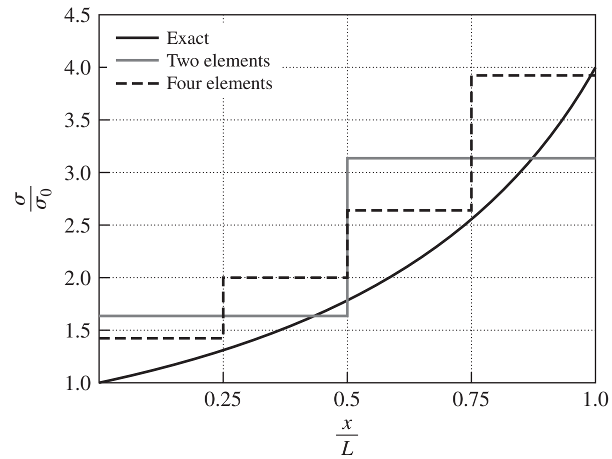


Figure 1.5
Comparison of the computed axial stress value in a tapered cylinder: $\sigma_0 = F/A_0$.

the derived variables are not necessarily continuous. In the limiting process of mesh refinement, the derived variables become closer and closer to continuity.

Our example shows how the finite element solution converges to a *known* exact solution (the exactness of the solution in this case is that of strength of materials theory). If we know the exact solution, we would not be applying the finite element method! So how do we assess the accuracy of a finite element solution for a problem with an unknown solution? The answer to this question is not simple. If we did not have the dashed line in Figure 1.3 representing the exact solution, we could still discern convergence to *a* solution. Convergence of a numerical method (such as the finite element method) is by no means assurance that the convergence is to the correct solution. A person using the finite element analysis technique must examine the solution analytically in terms of (1) numerical convergence, (2) reasonableness (does the result make sense?), (3) whether the physical laws of the problem are satisfied (is the structure in equilibrium? Does the heat output balance with the heat input?), and (4) whether the discontinuities in value of derived variables across element boundaries are reasonable. Many such questions must be posed and examined prior to accepting the results of a finite element analysis as representative of a correct solution useful for design purposes.

1.2.2 Comparison of Finite Element and Finite Difference Methods

The *finite difference* method is another numerical technique frequently used to obtain approximate solutions of problems governed by differential equations. Details of the technique are discussed in Chapter 7 in the context of transient heat transfer. The method is also illustrated in Chapter 10 for transient dynamic analysis of structures. Here, we present the basic concepts of the finite difference method for purposes of comparison.

The finite difference method is based on the definition of the derivative of a function $f(x)$ that is

$$\frac{\mathrm{d}f(x)}{\mathrm{d}x} = \lim_{\Delta x \to 0} \frac{f(x + \Delta x) - f(x)}{\Delta x} \tag{1.2}$$

where x is the independent variable. In the finite difference method, as implied by its name, derivatives are calculated via Equation 1.2 using small, but finite, values of Δx to obtain

$$\frac{\mathrm{d}f(x)}{\mathrm{d}x} \approx \frac{f(x + \Delta x) - f(x)}{\Delta x} \tag{1.3}$$

A differential equation such as

$$\frac{\mathrm{d}f}{\mathrm{d}x} + x = 0 \qquad 0 \le x \le 1 \tag{1.4}$$

is expressed as

$$\frac{f(x + \Delta x) - f(x)}{\Delta x} + x = 0 \tag{1.5}$$

in the finite difference method. Equation 1.5 can be rewritten as

$$f(x + \Delta x) = f(x) - x(\Delta x) \tag{1.6}$$

where we note that the equality must be taken as "approximately equals." From differential equation theory, we know that the solution of a first-order differential equation contains one constant of integration. The constant of integration must be determined such that one given condition (a boundary condition or initial condition) is satisfied. In the current example, we assume that the specified condition is $x(0) = A =$ constant. If we choose an *integration step* Δx to be a small, constant value (the integration step is not *required* to be constant), then we can write

$$x_{i+1} = x_i + \Delta x \qquad i = 0, N \tag{1.7}$$

where N is the total number of steps required to cover the domain. Equation 1.6 is then

$$f_{i+1} = f_i - x_i(\Delta x) \qquad f_0 = A \qquad i = 0, N \tag{1.8}$$

Equation 1.8 is known as a *recurrence relation* and provides an approximation to the value of the unknown function $f(x)$ at a number of discrete points in the domain of the problem.

To illustrate, Figure 1.6a shows the exact solution $f(x) = 1 - x^2/2$ and a finite difference solution obtained with $\Delta x = 0.1$. The finite difference solution is shown at the discrete points of function evaluation only. The manner of variation

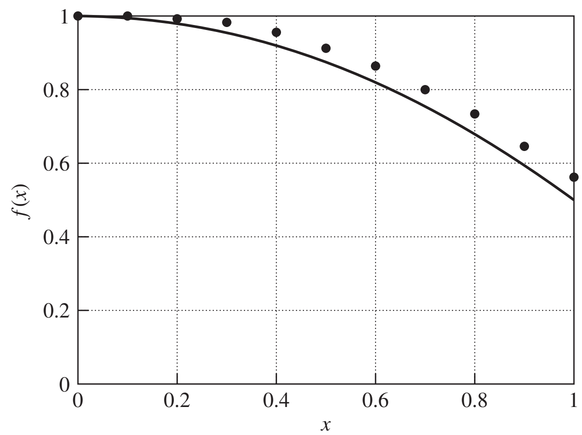


Figure 1.6
Comparison of the exact and finite difference solutions of Equation 1.4 with $f_0 = A = 1$.

of the function between the calculated points is not known in the finite difference method. One can, of course, linearly interpolate the values to produce an approximation to the curve of the exact solution but the manner of interpolation is not an a priori determination in the finite difference method.

To contrast the finite difference method with the finite element method, we note that, in the finite element method, the variation of the field variable in the physical domain is an integral part of the procedure. That is, based on the selected interpolation functions, the variation of the field variable throughout a finite element is specified as an integral part of the problem formulation. In the finite difference method, this is not the case: The field variable is computed at specified points only. The major ramification of this contrast is that derivatives (to a certain level) can be computed in the finite element approach, whereas the finite difference method provides data only on the variable itself. In a structural problem, for example, both methods provide displacement solutions, but the finite element solution can be used to directly compute strain components (first derivatives). To obtain strain data in the finite difference method requires additional considerations not inherent to the mathematical model.

There are also certain similarities between the two methods. The integration points in the finite difference method are analogous to the nodes in a finite element model. The variable of interest is explicitly evaluated at such points. Also, as the integration step (step size) in the finite difference method is reduced, the solution is expected to converge to the exact solution. This is similar to the expected convergence of a finite element solution as the mesh of elements is refined. In both cases, the refinement represents reduction of the mathematical model from finite to infinitesimal. And in both cases, differential equations are reduced to algebraic equations.

Probably the most descriptive way to contrast the two methods is to note that the finite difference method models the differential equation(s) of the problem and uses numerical integration to obtain the solution at discrete points. The finite element method models the entire domain of the problem and uses known physical principles to develop algebraic equations describing the approximate solutions. Thus, the finite difference method models differential equations while the finite element method can be said to more closely model the physical problem at hand. As will be observed in the remainder of this text, there are cases in which a combination of finite element and finite difference methods is very useful and efficient in obtaining solutions to engineering problems, particularly where dynamic (transient) effects are important.

1.3 A GENERAL PROCEDURE FOR FINITE ELEMENT ANALYSIS

Certain steps in formulating a finite element analysis of a physical problem are common to all such analyses, whether structural, heat transfer, fluid flow, or some other problem. These steps are embodied in commercial finite element software packages (some are mentioned in the following paragraphs) and are implicitly incorporated in this text, although we do not necessarily refer to the steps explicitly in the following chapters. The steps are described as follows.

1.3.1 Preprocessing

The preprocessing step is, quite generally, described as defining the model and includes

Define the geometric domain of the problem.
Define the element type(s) to be used (Chapter 6).
Define the material properties of the elements.
Define the geometric properties of the elements (length, area, and the like).
Define the element connectivities (mesh the model).
Define the physical constraints (boundary conditions).
Define the loadings.

The preprocessing (model definition) step is critical. In no case is there a better example of the computer-related axiom "garbage in, garbage out." A perfectly computed finite element solution is of absolutely no value if it corresponds to the wrong problem.

1.3.2 Solution

During the solution phase, finite element software assembles the governing algebraic equations in matrix form and computes the unknown values of the primary field variable(s). The computed values are then used by back substitution to compute additional, derived variables, such as reaction forces, element stresses, and heat flow.

As it is not uncommon for a finite element model to be represented by tens of thousands of equations, special solution techniques are used to reduce data storage requirements and computation time. For static, linear problems, a *wave front solver,* based on Gauss elimination (Appendix C), is commonly used. While a complete discussion of the various algorithms is beyond the scope of this text, the interested reader will find a thorough discussion in the Bathe book [1].

1.3.3 Postprocessing

Analysis and evaluation of the solution results is referred to as *postprocessing*. Postprocessor software contains sophisticated routines used for sorting, printing, and plotting selected results from a finite element solution. Examples of operations that can be accomplished include

Sort element stresses in order of magnitude.
Check equilibrium.
Calculate factors of safety.
Plot deformed structural shape.
Animate dynamic model behavior.
Produce color-coded temperature plots.

While solution data can be manipulated many ways in postprocessing, the most important objective is to apply sound engineering judgment in determining whether the solution results are physically reasonable.

1.4 BRIEF HISTORY OF THE FINITE ELEMENT METHOD

The mathematical roots of the finite element method dates back at least a half century. Approximate methods for solving differential equations using trial solutions are even older in origin. Lord Rayleigh [2] and Ritz [3] used trial functions (in our context, interpolation functions) to approximate solutions of differential equations. Galerkin [4] used the same concept for solution. The drawback in the earlier approaches, compared to the modern finite element method, is that the trial functions must apply over the *entire* domain of the problem of concern. While the Galerkin method provides a very strong basis for the finite element method (Chapter 5), not until the 1940s, when Courant [5] introduced the concept of piecewise-continuous functions in a subdomain, did the finite element method have its real start.

In the late 1940s, aircraft engineers were dealing with the invention of the jet engine and the needs for more sophisticated analysis of airframe structures to withstand larger loads associated with higher speeds. These engineers, without the benefit of modern computers, developed matrix methods of force analysis, collectively known as the *flexibility method,* in which the unknowns are the forces and the knowns are displacements. The finite element method, in its most often-used form, corresponds to the *displacement method,* in which the unknowns are system displacements in response to applied force systems. In this text, we adhere exclusively to the displacement method. As will be seen as we proceed, the term *displacement* is quite general in the finite element method and can represent physical displacement, temperature, or fluid velocity, for example. The term *finite element* was first used by Clough [6] in 1960 in the context of plane stress analysis and has been in common usage since that time.

During the decades of the 1960s and 1970s, the finite element method was extended to applications in plate bending, shell bending, pressure vessels, and general three-dimensional problems in elastic structural analysis [7–11] as well as to fluid flow and heat transfer [12, 13]. Further extension of the method to large deflections and dynamic analysis also occurred during this time period [14, 15]. An excellent history of the finite element method and detailed bibliography is given by Noor [16].

The finite element method is computationally intensive, owing to the required operations on very large matrices. In the early years, applications were performed using mainframe computers, which, at the time, were considered to be very powerful, high-speed tools for use in engineering analysis. During the 1960s, the finite element software code NASTRAN [17] was developed in conjunction with the space exploration program of the United States. NASTRAN was the first major finite element software code. It was, and still is, capable of hundreds of thousands of degrees of freedom (nodal field variable computations). In the years since the development of NASTRAN, many commercial software packages have been introduced for finite element analysis. Among these are ANSYS [18], ALGOR [19], and COSMOS/M [20]. In today's computational environment, most of these packages can be used on desktop computers and engineering workstations to obtain solutions to large problems in static and dynamic structural analysis, heat transfer, fluid flow, electromagnetics, and seismic response. In this text, we do not utilize or champion a particular code. Rather, we develop the fundamentals for understanding of finite element analysis to enable the reader to use such software packages with an educated understanding.

1.5 EXAMPLES OF FINITE ELEMENT ANALYSIS

We now present, briefly, a few examples of the types of problems that can be analyzed via the finite element method. Figure 1.7 depicts a rectangular region with a central hole. The area has been "meshed" with a finite element grid of two-dimensional elements assumed to have a constant thickness in the z direction. Note that the mesh of elements is irregular: The element shapes (triangles and quadrilaterals) and sizes vary. In particular, note that around the geometric discontinuity of the hole, the elements are of smaller size. This represents not only

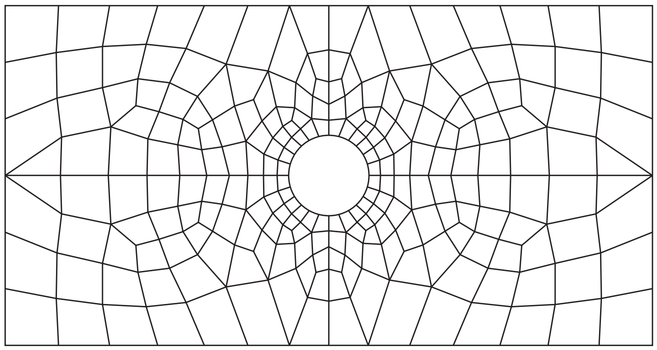


Figure 1.7
A mesh of finite elements over a rectangular region having a central hole.

an improvement in geometric accuracy in the vicinity of the discontinuity but also solution accuracy, as is discussed in subsequent chapters.

The geometry depicted in Figure 1.7 could represent the finite element model of several physical problems. For plane stress analysis, the geometry would represent a thin plate with a central hole subjected to edge loading in the plane depicted. In this case, the finite element solution would be used to examine stress concentration effects in the vicinity of the hole. The element mesh shown could also represent the case of fluid flow around a circular cylinder. In yet another application, the model shown could depict a heat transfer fin attached to a pipe (the hole) from which heat is transferred to the fin for dissipation to the surroundings. In each case, the formulation of the equations governing physical behavior of the elements in response to external influences is quite different.

Figure 1.8a shows a truss module that was at one time considered a building-block element for space station construction [21]. Designed to fold in accordion fashion into a small volume for transport into orbit, the module, when deployed, extends to overall dimensions 1.4 m × 1.4 m × 2.8 m. By attaching such modules end-to-end, a truss of essentially any length could be obtained. The structure was analyzed via the finite element method to determine the vibration characteristics as the number of modules, thus overall length, was varied. As the connections between the various structural members are pin or ball-and-socket joints, a simple axial tension-compression element (Chapter 2) was used in the model. The finite element model of one module was composed of 33 elements. A sample vibration shape of a five-module truss is shown in Figure 1.8b.

The truss example just described involves a rather large structure modeled by a small number of relatively large finite elements. In contrast, Figure 1.9 shows the finite element model of a very thin tube designed for use in heat

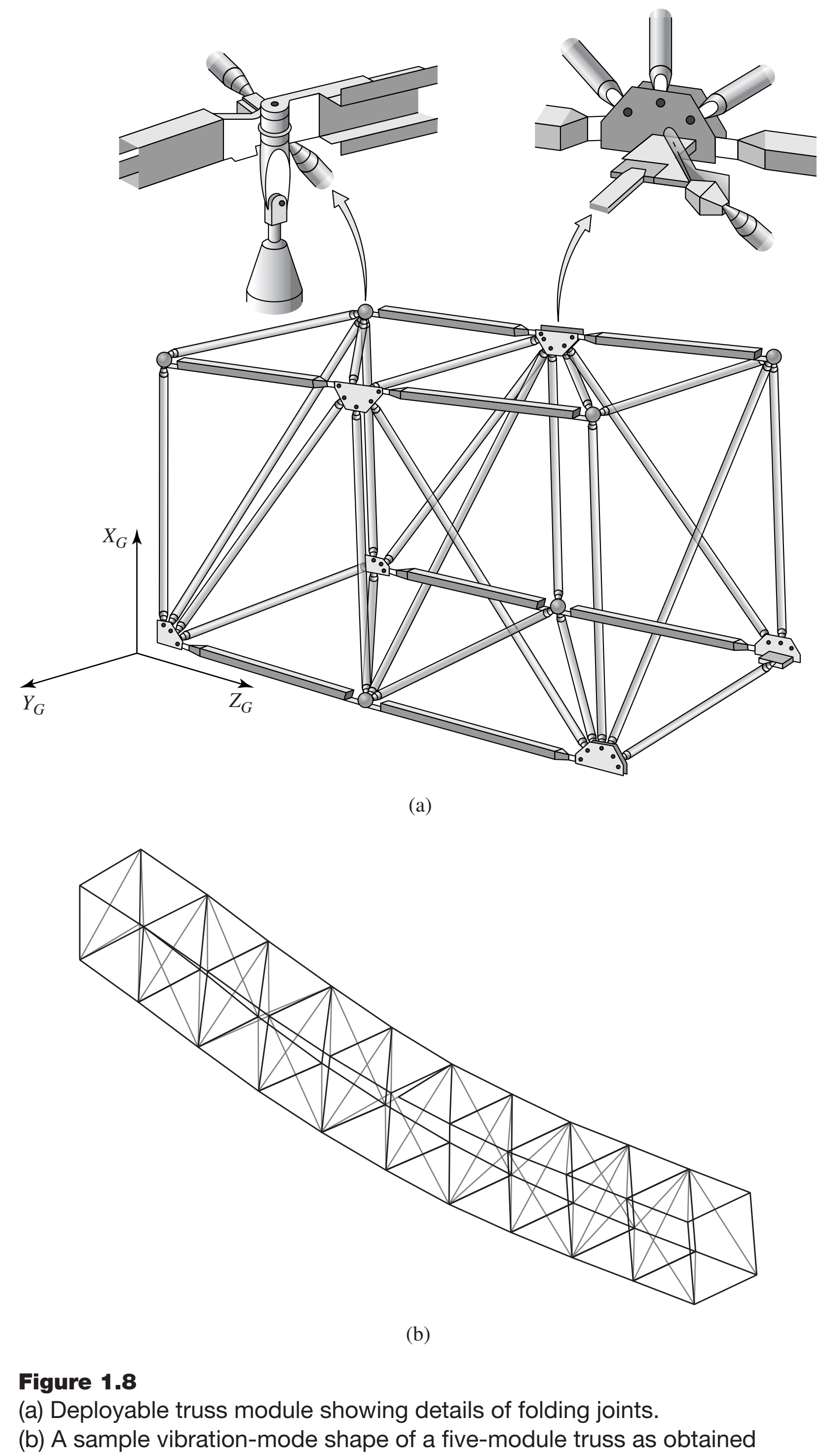


Figure 1.8
(a) Deployable truss module showing details of folding joints.
(b) A sample vibration-mode shape of a five-module truss as obtained via finite element analysis. *(Courtesy: AIAA)*

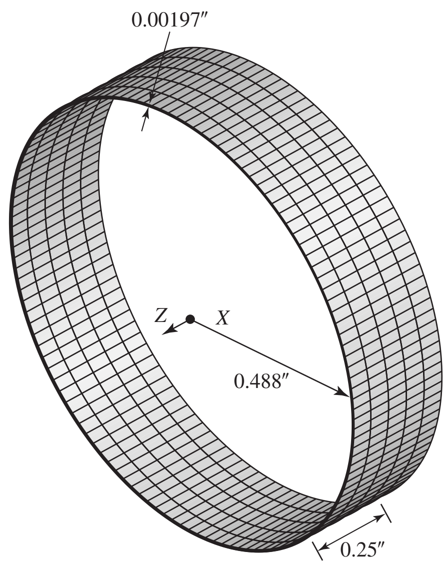
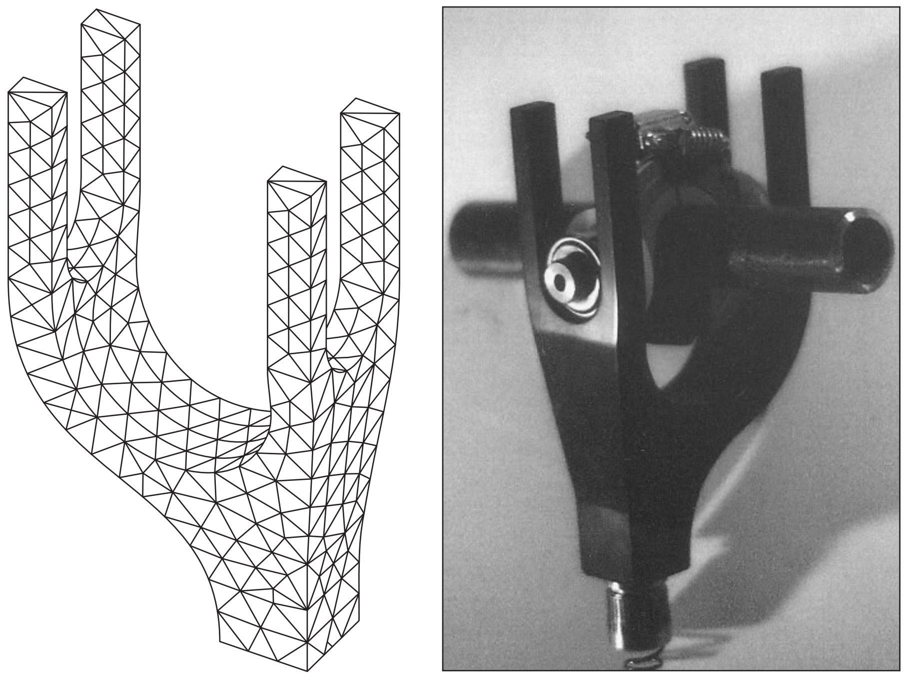


Figure 1.9
Finite element model of a thin-walled heat exchanger tube.

transfer in a spacecraft application. The tube has inside diameter of 0.976 in. and wall thickness 0.00197 in. and overall length 36 in. Materials considered for construction of the tube were copper and titanium alloys. Owing to the wall thickness, prototype tubes were found to be very fragile and difficult to handle without damage. The objectives of the finite element analysis were to examine the bending, torsional, and buckling loads allowable. The figure shows the finite element mesh used to model a section of the tube only 0.25 in. in length. This model contains 1920 three-dimensional solid elements, each having eight nodes with 3 degrees of freedom at each node. Such a large number of elements was required for a small structure in consideration of computational accuracy. The concern here was the so-called *aspect ratio* of the elements, as is defined and discussed in subsequent chapters.

As a final example, Figure 1.10a represents the finite element model of the main load-carrying component of a prosthetic device. The device is intended to be a hand attachment to an artificial arm. In use, the hand would allow a lower arm amputee to engage in weight lifting as part of a physical fitness program. The finite element model was used to determine the stress distribution in the component in terms of the range of weight loading anticipated, so as to properly size the component and select the material. Figure 1.10b shows a prototype of the completed hand design.



(a) (b)

Figure 1.10
(a) A finite element model of a prosthetic hand for weightlifting. (b) Completed prototype of a prosthetic hand, attached to a bar.
(Courtesy of Payam Sadat. All rights reserved.)

1.6 OBJECTIVES OF THE TEXT

I wrote *Fundamentals of Finite Element Analysis* for use in senior-level finite element courses in engineering programs. The majority of available textbooks on the finite element method are written for graduate-level courses. These texts are heavy on the theory of finite element analysis and rely on mathematical techniques (notably, *variational calculus*) that are not usually in the repertoire of undergraduate engineering students. Knowledge of advanced mathematical techniques is not required for successful use of this text. The prerequisite study is based on the undergraduate coursework common to most engineering programs: linear algebra, calculus through differential equations, and the usual series of statics, dynamics, and mechanics of materials. Although not required, prior study of fluid mechanics and heat transfer is helpful. Given this assumed background, the finite element method is developed on the basis of physical laws (equilibrium, conservation of mass, and the like), the principle of minimum potential energy (Chapter 2), and Galerkin's finite element method (introduced and developed in Chapter 5).

As the reader progresses through the text, he or she will discern that we cover a significant amount of finite element theory in addition to application examples. Given the availability of many powerful and sophisticated finite element software packages, why study the theory? The finite element method is a tool, and like any other tool, using it without proper instruction can be quite dangerous. My premise is that the proper instruction in this context includes understanding the basic theory underlying formulation of finite element models of physical problems. As stated previously, critical analysis of the results of a finite element model computation is essential, since those results may eventually become the basis for design. Knowledge of the theory is necessary for both proper modeling and evaluation of computational results.

REFERENCES

1. Bathe, K-J. *Finite Element Procedures*. Englewood Cliffs, NJ: Prentice-Hall, 1996.
2. Lord Rayleigh. "On the Theory of Resonance." *Transactions of the Royal Society (London)* A161 (1870).
3. Ritz, W. "Uber eine neue Methode zur Losung gewissen Variations-Probleme der mathematischen Physik." *J. Reine Angew. Math.* 135 (1909).
4. Galerkin, B. G. "Series Solution of Some Problems of Elastic Equilibrium of Rods and Plates" [in Russian]. *Vestn. Inzh. Tekh.* 19 (1915).
5. Courant, R. "Variational Methods for the Solution of Problems of Equilibrium and Vibrations." *Bulletin of the American Mathematical Society* 49 (1943).
6. Clough, R. W. "The Finite Element Method in Plane Stress Analysis." *Proceedings, American Society of Civil Engineers, Second Conference on Electronic Computation,* Pittsburgh, 1960.
7. Melosh, R. J. "A Stiffness Method for the Analysis of Thin Plates in Bending." *Journal of Aerospace Sciences* 28, no. 1 (1961).
8. Grafton, P. E., and D. R. Strome. "Analysis of Axisymmetric Shells by the Direct Stiffness Method." *Journal of the American Institute of Aeronautics and Astronautics* 1, no. 10 (1963).
9. Gallagher, R. H. "Analysis of Plate and Shell Structures." *Proceedings, Symposium on the Application of Finite Element Methods in Civil Engineering,* Vanderbilt University, Nashville, 1969.
10. Wilson, E. L. "Structural Analysis of Axisymmetric Solids." *Journal of the American Institute of Aeronautics and Astronautics* 3, (1965).
11. Melosh, R. J. "Structural Analysis of Solids." *Journal of the Structural Division,* Proceedings of the American Society of Civil Engineers, August 1963.
12. Martin, H. C. "Finite Element Analysis of Fluid Flows." *Proceedings of the Second Conference on Matrix Methods in Structural Mechanics,* Wright-Patterson Air Force Base, Kilborn, Ohio, October 1968.
13. Wilson, E. L., and R. E. Nickell. "Application of the Finite Element Method to Heat Conduction Analysis." *Nuclear Engineering Design* 4 (1966).

14. Turner, M. J., E. H. Dill, H. C. Martin, and R. J. Melosh. "Large Deflections of Structures Subjected to Heating and External Loads." *Journal of Aeronautical Sciences* 27 (1960).

15. Archer, J. S. "Consistent Mass Matrix Formulations for Structural Analysis Using Finite Element Techniques." *Journal of the American Institute of Aeronautics and Astronautics* 3, no. 10 (1965).

16. Noor, A. K. "Bibliography of Books and Monographs on Finite Element Technology." *Applied Mechanics Reviews* 44, no. 6 (1991).

17. MSC/NASTRAN. Lowell, MA: MacNeal-Schwindler Corp.

18. ANSYS. Houston, PA: Swanson Analysis Systems Inc.

19. ALGOR. Pittsburgh: Algor Interactive Systems.

20. COSMOS/M. Los Angeles: Structural Research and Analysis Corp.

21. Hutton, D. V. "Modal Analysis of a Deployable Truss Using the Finite Element Method." *Journal of Spacecraft and Rockets* 21, no. 5 (1984).

CHAPTER 2

Stiffness Matrices, Spring and Bar Elements

2.1 INTRODUCTION

The primary characteristics of a finite element are embodied in the element *stiffness matrix*. For a structural finite element, the stiffness matrix contains the geometric and material behavior information that indicates the resistance of the element to deformation when subjected to loading. Such deformation may include axial, bending, shear, and torsional effects. For finite elements used in nonstructural analyses, such as fluid flow and heat transfer, the term *stiffness matrix* is also used, since the matrix represents the resistance of the element to change when subjected to external influences.

This chapter develops the finite element characteristics of two relatively simple, one-dimensional structural elements, a linearly elastic spring and an elastic tension-compression member. These are selected as introductory elements because the behavior of each is relatively well-known from the commonly studied engineering subjects of statics and strength of materials. Thus, the “bridge” to the finite element method is not obscured by theories new to the engineering student. Rather, we build on known engineering principles to introduce finite element concepts. The linear spring and the tension-compression member (hereafter referred to as a *bar* element and also known in the finite element literature as a *spar, link,* or *truss* element) are also used to introduce the concept of *interpolation functions.* As mentioned briefly in Chapter 1, the basic premise of the finite element method is to describe the continuous variation of the field variable (in this chapter, physical displacement) in terms of discrete values at the finite element nodes. In the interior of a finite element, as well as along the boundaries (applicable to two- and three-dimensional problems), the field variable is described via interpolation functions (Chapter 6) that must satisfy prescribed conditions.

Finite element analysis is based, dependent on the type of problem, on several mathematic/physical principles. In the present introduction to the method, we present several such principles applicable to finite element analysis. First, and foremost, for spring and bar systems, we utilize the principle of static equilibrium but—and this is essential—we include *deformation* in the development; that is, we are not dealing with rigid body mechanics. For extension of the finite element method to more complicated elastic structural systems, we also state and apply the first theorem of Castigliano [1] and the more widely used principle of minimum potential energy [2]. Castigliano's first theorem, in the form presented, may be new to the reader. The first theorem is the counterpart of Castigliano's second theorem, which is more often encountered in the study of elementary strength of materials [3]. Both theorems relate displacements and applied forces to the equilibrium conditions of a mechanical system in terms of mechanical energy. The use here of Castigliano's first theorem is for the distinct purpose of introducing the concept of minimum potential energy without resort to the higher mathematic principles of the calculus of variations, which is beyond the mathematical level intended for this text.

2.2 LINEAR SPRING AS A FINITE ELEMENT

A linear elastic spring is a mechanical device capable of supporting axial loading only and constructed such that, over a reasonable operating range (meaning extension or compression beyond undeformed length), the elongation or contraction of the spring is directly proportional to the applied axial load. The constant of proportionality between deformation and load is referred to as the *spring constant, spring rate,* or *spring stiffness* [4], generally denoted as k, and has units of force per unit length. Formulation of the linear spring as a finite element is accomplished with reference to Figure 2.1a. As an elastic spring supports axial loading only, we select an *element coordinate system* (also known as a *local* coordinate system) as an x axis oriented along the length of the spring, as shown. The element coordinate system is embedded in the element and chosen, by geometric convenience, for simplicity in describing element behavior. The element

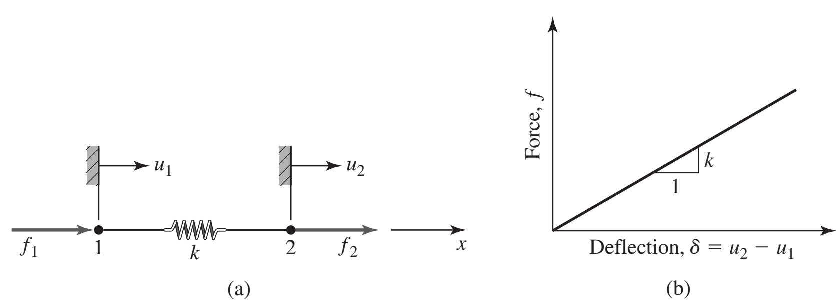


Figure 2.1
(a) Linear spring element with nodes, nodal displacements, and nodal forces.
(b) Load-deflection curve.

or local coordinate system is contrasted with the *global* coordinate system. The global coordinate system is that system in which the behavior of a complete structure is to be described. By *complete structure* is meant the assembly of many finite elements (at this point, several springs) for which we desire to compute response to loading conditions. In this chapter, we deal with cases in which the local and global coordinate systems are essentially the same except for translation of origin. In two- and three-dimensional cases, however, the distinctions are quite different and require mathematical rectification of element coordinate systems to a common basis. The common basis is the global coordinate system.

Returning attention to Figure 2.1a, the ends of the spring are the *nodes* and the nodal displacements are denoted by u_1 and u_2 and are shown in the positive sense. If these nodal displacements are known, the total elongation or contraction of the spring is known as is the *net force* in the spring. At this point in our development, we require that forces be applied to the element only at the nodes (distributed forces are accommodated for other element types later), and these are denoted as f_1 and f_2 and are also shown in the positive sense.

Assuming that both the nodal displacements are zero when the spring is undeformed, the net spring deformation is given by

$$\delta = u_2 - u_1 \tag{2.1}$$

and the resultant axial force in the spring is

$$f = k\delta = k(u_2 - u_1) \tag{2.2}$$

as is depicted in Figure 2.1b.

For equilibrium, $f_1 + f_2 = 0$ or $f_1 = -f_2$, and we can rewrite Equation 2.2 in terms of the applied nodal forces as

$$f_1 = -k(u_2 - u_1) \tag{2.3a}$$

$$f_2 = k(u_2 - u_1) \tag{2.3b}$$

which can be expressed in matrix form (see Appendix A for a review of matrix algebra) as

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \tag{2.4}$$

or

$$[k_e]\{u\} = \{f\} \tag{2.5}$$

where

$$[k_e] = \begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \tag{2.6}$$

is defined as the element stiffness matrix in the element coordinate system (or local system), $\{u\}$ is the column matrix (vector) of nodal displacements, and $\{f\}$ is the column matrix (vector) of element nodal forces. (In subsequent chapters, the matrix notation is used extensively. A general matrix is designated by brackets [ ] and a column matrix (vector) by braces { }.)

Equation 2.6 shows that the element stiffness matrix for the linear spring element is a 2×2 matrix. This corresponds to the fact that the element exhibits two nodal displacements (or degrees of freedom) and that the two displacements are not independent (that is, the body is continuous and elastic). Furthermore, the matrix is symmetric. A symmetric matrix has off-diagonal terms such that $k_{ij} = k_{ji}$. Symmetry of the stiffness matrix is indicative of the fact that the body is linearly elastic and each displacement is related to the other by the same physical phenomenon. For example, if a force F (positive, tensile) is applied at node 2 with node 1 held fixed, the *relative* displacement of the two nodes is the same as if the force is applied *symmetrically* (negative, tensile) at node 1 with node 2 fixed. (Counterexamples to symmetry are seen in heat transfer and fluid flow analyses in Chapters 7 and 8.) As will be seen as more complicated structural elements are developed, this is a general result: An element exhibiting N degrees of freedom has a corresponding $N \times N$, symmetric stiffness matrix.

Next consider solution of the system of equations represented by Equation 2.4. In general, the nodal forces are prescribed and the objective is to solve for the unknown nodal displacements. Formally, the solution is represented by

$$\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = [k_e]^{-1} \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \tag{2.7}$$

where $[k_e]^{-1}$ is the inverse of the element stiffness matrix. However, this inverse matrix does not exist, since the determinant of the element stiffness matrix is identically zero. Therefore, the element stiffness matrix is *singular,* and this also proves to be a general result in most cases. The physical significance of the singular nature of the element stiffness matrix is found by reexamination of Figure 2.1a, which shows that no displacement constraint whatever has been imposed on motion of the spring element; that is, the spring is not connected to any physical object that would prevent or limit motion of either node. With no constraint, it is not possible to solve for the nodal displacements individually. Instead, only the *difference* in nodal displacements can be determined, as this difference represents the elongation or contraction of the spring element owing to elastic effects. As discussed in more detail in the general formulation of interpolation functions (Chapter 6) and structural dynamics (Chapter 10), a properly formulated finite element must allow for constant value of the field variable. In the example at hand, this means rigid body motion. We can see the rigid body motion capability in terms of a single spring (element) and in the context of several connected elements. For a single, unconstrained element, if arbitrary forces are applied at each node, the spring not only deforms axially but also undergoes acceleration according to Newton's second law. Hence, there exists not only deformation but overall motion. If, in a connected system of spring elements, the overall system response is such that nodes 1 and 2 of a particular element displace the same amount, there is no elastic deformation of the spring and therefore no elastic force in the spring. This physical situation must be included in the element formulation. The capability is indicated mathematically by singularity of the element stiffness matrix. As the stiffness matrix is formulated on the basis of *deformation* of the element, we cannot expect to compute nodal displacements if there is no deformation of the element.

Equation 2.7 indicates the mathematical operation of inverting the stiffness matrix to obtain solutions. In the context of an individual element, the singular nature of an element stiffness matrix precludes this operation, as the inverse of a singular matrix does not exist. As is illustrated profusely in the remainder of the text, the general solution of a finite element problem, in a global, as opposed to element, context, involves the solution of equations of the form of Equation 2.5. For realistic finite element models, which are of huge dimension in terms of the matrix order ($N \times N$) involved, computing the inverse of the stiffness matrix is a very inefficient, time-consuming operation, which should not be undertaken except for the very simplest of systems. Other, more-efficient solution techniques are available, and these are discussed subsequently. (Many of the end-of-chapter problems included in this text are of small order and can be efficiently solved via matrix inversion using "spreadsheet" software functions or software such as MATLAB.)

2.2.1 System Assembly in Global Coordinates

Derivation of the element stiffness matrix for a spring element was based on equilibrium conditions. The same procedure can be applied to a connected system of spring elements by writing the equilibrium equation for each node. However, rather than drawing free-body diagrams of each node and formally writing the equilibrium equations, the nodal equilibrium equations can be obtained more efficiently by considering the effect of each element separately and adding the element force contribution to each nodal equation. The process is described as "assembly," as we take individual stiffness components and "put them together" to obtain the system equations. To illustrate, via a simple example, the assembly of element characteristics into *global* (or *system*) equations, we next consider the system of two linear spring elements connected as shown in Figure 2.2.

For generality, it is assumed that the springs have different spring constants k_1 and k_2. The nodes are numbered 1, 2, and 3 as shown, with the springs sharing node 2 as the physical connection. Note that these are *global* node numbers. The *global* nodal displacements are identified as U_1, U_2, and U_3, where the upper case is used to indicate that the quantities represented are *global* or *system* displacements as opposed to individual element displacements. Similarly, applied nodal

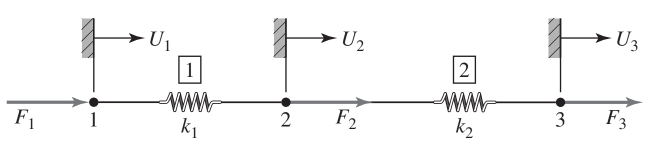


Figure 2.2 System of two springs with node numbers, element numbers, nodal displacements, and nodal forces.

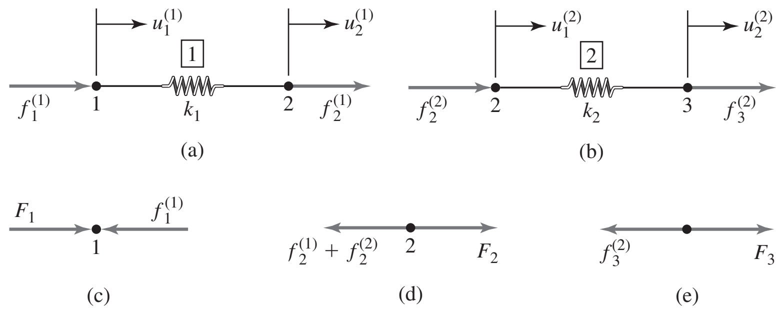


Figure 2.3 Free-body diagrams of elements and nodes for the two-element system of Figure 2.2.

forces are F_1, F_2, and F_3. Assuming the system of two spring elements to be in equilibrium, we examine free-body diagrams of the springs individually (Figure 2.3a and 2.3b) and express the equilibrium conditions for each spring, using Equation 2.4, as

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} u_1^{(1)} \\ u_2^{(1)} \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix} \tag{2.8a}$$

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} u_1^{(2)} \\ u_2^{(2)} \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix} \tag{2.8b}$$

where the superscript is element number.

To begin "assembling" the equilibrium equations describing the behavior of the system of two springs, the displacement *compatibility conditions,* which relate element displacements to system displacements, are written as

$$u_1^{(1)} = U_1 \qquad u_2^{(1)} = U_2 \qquad u_1^{(2)} = U_2 \qquad u_2^{(2)} = U_3 \tag{2.9}$$

The compatibility conditions state the physical fact that the springs are connected at node 2, remain connected at node 2 after deformation, and hence, must have the same nodal displacement at node 2. Thus, element-to-element displacement continuity is enforced at nodal connections. Substituting Equations 2.9 into Equations 2.8, we obtain

$$\begin{bmatrix} k_1 & -k_1 \\ -k_1 & k_1 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix} \tag{2.10a}$$

and

$$\begin{bmatrix} k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix} \tag{2.10b}$$

Here, we use the notation $f_i^{(j)}$ to represent the force exerted on element j at node i.

Equation 2.10 is the equilibrium equations for each spring element expressed in terms of the specified global displacements. In this form, the equations clearly show that the elements are physically connected at node 2 and have the same displacement U_2 at that node. These equations are not yet amenable to direct combination, as the displacement vectors are not the same. We expand both matrix equations to 3×3 as follows (while formally expressing the facts that element 1 is not connected to node 3 and element 2 is not connected to node 1):

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 & 0 \\ 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ 0 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \\ 0 \end{Bmatrix} \tag{2.11}$$

$$\begin{bmatrix} 0 & 0 & 0 \\ 0 & k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix} \tag{2.12}$$

The addition of Equations 2.11 and 2.12 yields

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} + f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix} \tag{2.13}$$

Next, we refer to the free-body diagrams of each of the three nodes depicted in Figure 2.3c, 2.3d, and 2.3e. The equilibrium conditions for nodes 1, 2, and 3 show that

$$f_1^{(1)} = F_1 \qquad f_2^{(1)} + f_2^{(2)} = F_2 \qquad f_3^{(2)} = F_3 \tag{2.14}$$

respectively. Substituting into Equation 2.13, we obtain the final result:

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \end{Bmatrix} \tag{2.15}$$

which is of the form $[K]\{U\} = \{F\}$, similar to Equation 2.5. However, Equation 2.15 represents the equations governing the *system* composed of two connected spring elements. By direct consideration of the equilibrium conditions, we obtain the system stiffness matrix $[K]$ (note use of upper case) as

$$[K] = \begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \tag{2.16}$$

Note that the system stiffness matrix is (1) symmetric, as is the case with all linear systems referred to orthogonal coordinate systems; (2) singular, since no constraints are applied to prevent rigid body motion of the system; and (3) the system matrix is simply a superposition of the individual element stiffness matrices with proper assignment of element nodal displacements and associated stiffness coefficients to system nodal displacements. The superposition procedure is formalized in the context of frame structures in the following paragraphs.

EXAMPLE 2.1

Consider the two element system depicted in Figure 2.2 given that

Node 1 is attached to a fixed support, yielding the displacement constraint $U_1 = 0$.
$k_1 = 50$ lb./in., $k_2 = 75$ lb./in., $F_2 = F_3 = 75$ lb.

for these conditions determine nodal displacements U_2 and U_3.

■ **Solution**

Substituting the specified values into Equation 2.15 yields

$$\begin{bmatrix} 50 & -50 & 0 \\ -50 & 125 & -75 \\ 0 & -75 & 75 \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 75 \\ 75 \end{Bmatrix}$$

and we note that, owing to the constraint of zero displacement at node 1, nodal force F_1 becomes an unknown reaction force. Formally, the first algebraic equation represented in this matrix equation becomes

$$-50U_2 = F_1$$

and this is known as a *constraint equation,* as it represents the equilibrium condition of a node at which the displacement is constrained. The second and third equations become

$$\begin{bmatrix} 125 & -75 \\ -75 & 75 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} 75 \\ 75 \end{Bmatrix}$$

which can be solved to obtain $U_2 = 3$ in. and $U_3 = 4$ in. Note that the matrix equations governing the unknown displacements are obtained by simply striking out the first row and column of the 3×3 matrix system, since the constrained displacement is zero. Hence, the constraint does not affect the values of the *active* displacements (we use the term *active* to refer to displacements that are unknown and must be computed). Substitution of the calculated values of U_2 and U_3 into the constraint equation yields the value $F_1 = -150$ lb., which value is clearly in equilibrium with the applied nodal forces of 75 lb. each. We also illustrate element equilibrium by writing the equations for each element as

$$\begin{bmatrix} 50 & -50 \\ -50 & 50 \end{bmatrix} \begin{Bmatrix} 0 \\ 3 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix} = \begin{Bmatrix} -150 \\ 150 \end{Bmatrix} \text{lb.} \qquad \text{for element 1}$$

$$\begin{bmatrix} 75 & -75 \\ -75 & 75 \end{bmatrix} \begin{Bmatrix} 3 \\ 4 \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix} = \begin{Bmatrix} -75 \\ 75 \end{Bmatrix} \text{lb.} \qquad \text{for element 2}$$

Example 2.1 illustrates the general procedure for solution of finite element models: Formulate the system equilibrium equations, apply the specified constraint conditions, solve the reduced set of equations for the "active" displacements, and substitute the computed displacements into the constraint equations to obtain the unknown reactions. While not directly applicable for the spring element, for more general finite element formulations, the computed displacements are also substituted into the strain relations to obtain element strains, and the strains are, in turn, substituted into the applicable stress-strain equations to obtain element stress values.

EXAMPLE 2.2

Figure 2.4a depicts a system of three linearly elastic springs supporting three equal weights W suspended in a vertical plane. Treating the springs as finite elements, determine the vertical displacement of each weight.

■ Solution

To treat this as a finite element problem, we assign node and element numbers as shown in Figure 2.4b and ignore, for the moment, that displacement U_1 is known to be zero by the fixed support constraint. Per Equation 2.6, the stiffness matrix of each element is (preprocessing)

$$[k^{(1)}] = \begin{bmatrix} 3k & -3k \\ -3k & 3k \end{bmatrix}$$

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

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

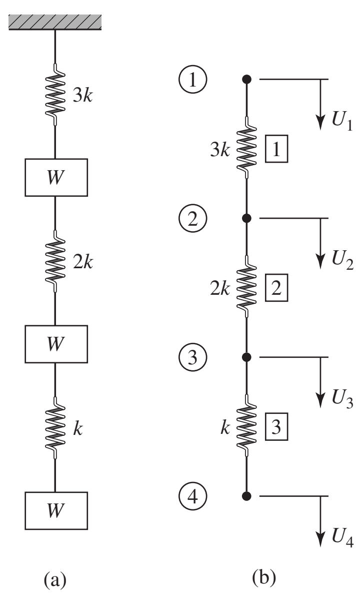


Figure 2.4 Example 2.2: elastic spring supporting weights.

The element-to-global displacement relations are

$$u_1^{(1)} = U_1 \qquad u_2^{(1)} = u_1^{(2)} = U_2 \qquad u_2^{(2)} = u_1^{(3)} = U_3 \qquad u_2^{(3)} = U_4$$

Proceeding as in the previous example, we then write the individual element equations as

$$\begin{bmatrix} 3k & -3k & 0 & 0 \\ -3k & 3k & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \\ 0 \\ 0 \end{Bmatrix} \tag{1}$$

$$\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 2k & -2k & 0 \\ 0 & -2k & 2k & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ f_1^{(2)} \\ f_2^{(2)} \\ 0 \end{Bmatrix} \tag{2}$$

$$\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 0 \\ 0 & 0 & k & -k \\ 0 & 0 & -k & k \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} 0 \\ 0 \\ f_1^{(3)} \\ f_2^{(3)} \end{Bmatrix} \tag{3}$$

Adding Equations 1–3, we obtain

$$k \begin{bmatrix} 3 & -3 & 0 & 0 \\ -3 & 5 & -2 & 0 \\ 0 & -2 & 3 & -1 \\ 0 & 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ W \\ W \\ W \end{Bmatrix} \tag{4}$$

where we utilize the fact that the sum of the element forces at each node must equal the applied force at that node and, at node 1, the force is an unknown reaction.

Applying the displacement constraint $U_1 = 0$ (*this is also preprocessing*), we obtain

$$-3kU_2 = F_1 \tag{5}$$

as the constraint equation and the matrix equation

$$k \begin{bmatrix} 5 & -2 & 0 \\ -2 & 3 & -1 \\ 0 & -1 & 1 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} W \\ W \\ W \end{Bmatrix} \tag{6}$$

for the active displacements. Again note that Equation 6 is obtained by eliminating the constraint equation from 4 corresponding to the prescribed zero displacement.

Simultaneous solution (*the solution step*) of the algebraic equations represented by Equation 6 yields the displacements as

$$U_2 = \frac{W}{k} \qquad U_3 = \frac{2W}{k} \qquad U_4 = \frac{3W}{k}$$

and Equation 5 gives the reaction force as

$$F_1 = -3W$$

(This is *postprocessing*.)

Note that the solution is exactly that which would be obtained by the usual statics equations. Also note the general procedure as follows:

Formulate the individual element stiffness matrices.

Write the element to global displacement relations.

Assemble the global equilibrium equation in matrix form.

Reduce the matrix equations according to specified constraints.

Solve the system of equations for the unknown nodal displacements (primary variables).

Solve for the reaction forces (secondary variable) by back-substitution.

EXAMPLE 2.3

Figure 2.5 depicts a system of three linear spring elements connected as shown. The node and element numbers are as indicated. Node 1 is fixed to prevent motion, and node 3 is given a specified displacement δ as shown. Forces $F_2 = -F$ and $F_4 = 2F$ are applied at nodes 2 and 4. Determine the displacement of each node and the force required at node 3 for the specified conditions.

■ Solution

This example includes a *nonhomogeneous* boundary condition. In previous examples, the boundary conditions were represented by zero displacements. In this example, we have both a zero (homogeneous) and a specified nonzero (nonhomogeneous) displacement condition. The algebraic treatment must be different as follows. The system equilibrium equations are expressed in matrix form (Problem 2.6) as

$$\begin{bmatrix} k & -k & 0 & 0 \\ -k & 4k & -3k & 0 \\ 0 & -3k & 5k & -2k \\ 0 & 0 & -2k & 2k \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ -F \\ F_3 \\ 2F \end{Bmatrix}$$

Substituting the specified conditions $U_1 = 0$ and $U_3 = \delta$ results in the system of equations

$$\begin{bmatrix} k & -k & 0 & 0 \\ -k & 4k & -3k & 0 \\ 0 & -3k & 5k & -2k \\ 0 & 0 & -2k & 2k \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \\ \delta \\ U_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ -F \\ F_3 \\ 2F \end{Bmatrix}$$

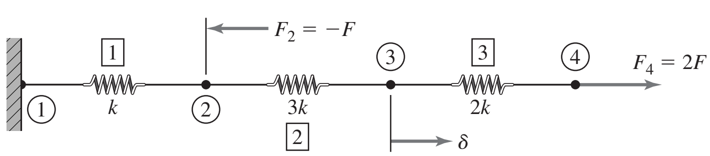


Figure 2.5 Example 2.3: Three-element system with specified nonzero displacement at node 3.

Since $U_1 = 0$, we remove the first row and column to obtain

$$\begin{bmatrix} 4k & -3k & 0 \\ -3k & 5k & -2k \\ 0 & -2k & 2k \end{bmatrix} \begin{Bmatrix} U_2 \\ \delta \\ U_4 \end{Bmatrix} = \begin{Bmatrix} -F \\ F_3 \\ 2F \end{Bmatrix}$$

as the system of equations governing displacements U_2 and U_4 and the unknown nodal force F_3. This last set of equations clearly shows that we cannot simply strike out the row and column corresponding to the *nonzero* specified displacement δ because it appears in the equations governing the active displacements. To illustrate a general procedure, we rewrite the last matrix equation as

$$\begin{bmatrix} 5k & -3k & -2k \\ -3k & 4k & 0 \\ -2k & 0 & 2k \end{bmatrix} \begin{Bmatrix} \delta \\ U_2 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} F_3 \\ -F \\ 2F \end{Bmatrix}$$

Next, we formally partition the stiffness matrix and write

$$\begin{bmatrix} 5k & -3k & -2k \\ -3k & 4k & 0 \\ -2k & 0 & 2k \end{bmatrix} \begin{Bmatrix} \delta \\ U_2 \\ U_4 \end{Bmatrix} = \begin{bmatrix} [K_{\delta\delta}] & [K_{\delta U}] \\ [K_{U\delta}] & [K_{UU}] \end{bmatrix} \begin{Bmatrix} \{\delta\} \\ \{U\} \end{Bmatrix} = \begin{Bmatrix} \{F_\delta\} \\ \{F_U\} \end{Bmatrix}$$

with

$$[K_{\delta\delta}] = [5k]$$

$$[K_{\delta U}] = [-3k \quad -2k]$$

$$[K_{U\delta}] = [K_{\delta U}]^T = \begin{bmatrix} -3k \\ -2k \end{bmatrix}$$

$$[K_{UU}] = \begin{bmatrix} 4k & 0 \\ 0 & 2k \end{bmatrix}$$

$$\{\delta\} = \{\delta\}$$

$$\{U\} = \begin{Bmatrix} U_2 \\ U_4 \end{Bmatrix}$$

$$\{F_\delta\} = \{F_3\}$$

$$\{F_U\} = \begin{Bmatrix} -F \\ 2F \end{Bmatrix}$$

From the second “row” of the partitioned matrix equations, we have

$$[K_{U\delta}]\{\delta\} + [K_{UU}]\{U\} = \{F_U\}$$

and this can be solved for the unknown displacements to obtain

$$\{U\} = [K_{UU}]^{-1}(\{F\} - [K_{U\delta}]\{\delta\})$$

provided that $[K_{UU}]^{-1}$ exists. Since the constraints have been applied correctly, this inverse does exist and is given by

$$[K_{UU}]^{-1} = \begin{bmatrix} \dfrac{1}{4k} & 0 \\ 0 & \dfrac{1}{2k} \end{bmatrix}$$

Substituting, we obtain the unknown displacements as

$$\{U\} = \begin{Bmatrix} U_2 \\ U_4 \end{Bmatrix} = \begin{bmatrix} \dfrac{1}{4k} & 0 \\ 0 & \dfrac{1}{2k} \end{bmatrix} \begin{Bmatrix} -F + 3k\delta \\ 2F + 2k\delta \end{Bmatrix} = \begin{Bmatrix} -\dfrac{F}{4k} + \dfrac{3\delta}{4} \\ \dfrac{F}{k} + \delta \end{Bmatrix}$$

The required force at node 3 is obtained by substitution of the displacement into the upper partition to obtain

$$F_3 = -\frac{5}{4}F + \frac{3}{4}k\delta$$

Finally, the reaction force at node 1 is

$$F_1 = -kU_2 = \frac{F}{4} - \frac{3}{4}k\delta$$

As a check on the results, we substitute the computed and prescribed displacements into the individual element equations to insure that equilibrium is satisfied.

Element 1

$$\begin{bmatrix} k & -k \\ -k & k \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \end{Bmatrix} = \begin{Bmatrix} -kU_2 \\ kU_2 \end{Bmatrix} = \begin{Bmatrix} f_1^{(1)} \\ f_2^{(1)} \end{Bmatrix}$$

which shows that the nodal forces on element 1 are equal and opposite as required for equilibrium.

Element 2

$$\begin{bmatrix} 3k & -3k \\ -3k & 3k \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{bmatrix} 3k & -3k \\ -3k & 3k \end{bmatrix} \begin{Bmatrix} -\dfrac{F}{4k} + \dfrac{3}{4}\delta \\ \delta \end{Bmatrix}$$

$$= \begin{Bmatrix} -\dfrac{3F}{4k} - \dfrac{3}{4}k\delta \\ \dfrac{3F}{4k} + \dfrac{3}{4}k\delta \end{Bmatrix} = \begin{Bmatrix} f_2^{(2)} \\ f_3^{(2)} \end{Bmatrix}$$

which also verifies equilibrium.

Element 3

$$\begin{bmatrix} 2k & -2k \\ -2k & 2k \end{bmatrix} \begin{Bmatrix} U_3 \\ U_4 \end{Bmatrix} = \begin{bmatrix} 2k & -2k \\ -2k & 2k \end{bmatrix} \begin{Bmatrix} \delta \\ \dfrac{F}{k} + \delta \end{Bmatrix} = \begin{Bmatrix} -2F \\ 2F \end{Bmatrix} = \begin{Bmatrix} f_3^{(3)} \\ f_4^{(3)} \end{Bmatrix}$$

Therefore element 3 is in equilibrium as well.

2.3 ELASTIC BAR, SPAR/LINK/TRUSS ELEMENT

While the linear elastic spring serves to introduce the concept of the stiffness matrix, the usefulness of such an element in finite element analysis is rather limited. Certainly, springs are used in machinery in many cases and the availability of a finite element representation of a linear spring is quite useful in such cases. The spring element is also often used to represent the elastic nature of supports for more complicated systems. A more generally applicable, yet similar, element is an elastic bar subjected to axial forces only. This element, which we simply call a *bar element,* is particularly useful in the analysis of both two- and three-dimensional frame or truss structures. Formulation of the finite element characteristics of an elastic bar element is based on the following assumptions:

1. The bar is geometrically straight.
2. The material obeys Hooke's law.
3. Forces are applied only at the ends of the bar.
4. The bar supports axial loading only; bending, torsion, and shear are not transmitted to the element via the nature of its connections to other elements.

The last assumption, while quite restrictive, is not impractical; this condition is satisfied if the bar is connected to other structural members via pins (2-D) or ball-and-socket joints (3-D). Assumptions 1 and 4, in combination, show that this is inherently a one-dimensional element, meaning that the elastic displacement of any point along the bar can be expressed in terms of a single independent variable. As will be seen, however, the bar element can be used in modeling both two- and three-dimensional structures. The reader will recognize this element as the familiar two-force member of elementary statics, meaning, for equilibrium, the forces exerted on the ends of the element must be colinear, equal in magnitude, and opposite in sense.

Figure 2.6 depicts an elastic bar of length L to which is affixed a uniaxial coordinate system x with its origin arbitrarily placed at the left end. This is the *element* coordinate system or reference frame. Denoting axial displacement at any position along the length of the bar as $u(x)$, we define nodes 1 and 2 at each end as shown and introduce the nodal displacements $u_1 = u(x = 0)$ and $u_2 = u(x = L)$. Thus, we have the continuous field variable $u(x)$, which is to be expressed (approximately) in terms of two nodal variables u_1 and u_2. To accomplish this discretization, we assume the existence of *interpolation* functions $N_1(x)$ and $N_2(x)$ (also known as *shape* or *blending* functions) such that

$$u(x) = N_1(x)u_1 + N_2(x)u_2 \tag{2.17}$$

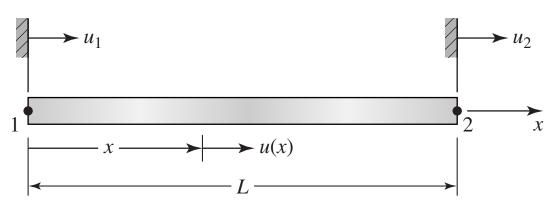


Figure 2.6 A bar (or truss) element with element coordinate system and nodal displacement notation.

(It must be emphasized that, although an equality is indicated by Equation 2.17, the relation, for finite elements in general, is an approximation. For the bar element, the relation, in fact, is exact.) To determine the interpolation functions, we require that the boundary values of $u(x)$ (the nodal displacements) be identically satisfied by the discretization such that

$$u(x = 0) = u_1 \qquad u(x = L) = u_2 \tag{2.18}$$

Equations 2.17 and 2.18 lead to the following boundary (nodal) conditions:

$$N_1(0) = 1 \qquad N_2(0) = 0 \tag{2.19}$$

$$N_1(L) = 0 \qquad N_2(L) = 1 \tag{2.20}$$

which must be satisfied by the interpolation functions. It is required that the displacement expression, Equation 2.17, satisfy the end (nodal) conditions identically, since the nodes will be the connection points between elements and the displacement continuity conditions are enforced at those connections. As we have two conditions that must be satisfied by each of two one-dimensional functions, the simplest forms for the interpolation functions are polynomial forms:

$$N_1(x) = a_0 + a_1 x \tag{2.21}$$

$$N_2(x) = b_0 + b_1 x \tag{2.22}$$

where the polynomial coefficients are to be determined via satisfaction of the boundary (nodal) conditions. We note here that any number of mathematical forms of the interpolation functions could be assumed while satisfying the required conditions. The reasons for the linear form is explained in detail in Chapter 6.

Application of conditions represented by Equation 2.19 yields $a_0 = 1$, $b_0 = 0$ while Equation 2.20 results in $a_1 = -(1/L)$ and $b_1 = x/L$. Therefore, the interpolation functions are

$$N_1(x) = 1 - x/L \tag{2.23}$$

$$N_2(x) = x/L \tag{2.24}$$

and the continuous displacement function is represented by the discretization

$$u(x) = (1 - x/L)u_1 + (x/L)u_2 \tag{2.25}$$

As will be found most convenient subsequently, Equation 2.25 can be expressed in matrix form as

$$u(x) = [N_1(x) \quad N_2(x)]\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = [N]\{u\} \tag{2.26}$$

where $[N]$ is the row matrix of interpolation functions and $\{u\}$ is the column matrix (vector) of nodal displacements.

Having expressed the displacement field in terms of the nodal variables, the task remains to determine the relation between the nodal displacements and applied forces to obtain the stiffness matrix for the bar element. Recall from elementary strength of materials that the deflection δ of an elastic bar of length L and uniform cross-sectional area A when subjected to axial load P is given by

$$\delta = \frac{PL}{AE} \tag{2.27}$$

where E is the modulus of elasticity of the material. Using Equation 2.27, we obtain the equivalent spring constant of an elastic bar as

$$k = \frac{P}{\delta} = \frac{AE}{L} \tag{2.28}$$

and could, by analogy with the linear elastic spring, immediately write the stiffness matrix as Equation 2.6. While the result is exactly correct, we take a more general approach to illustrate the procedures to be used with more complicated element formulations.

Ultimately, we wish to compute the nodal displacements given some loading condition on the element. To obtain the necessary equilibrium equations relating the displacements to applied forces, we proceed from displacement to strain, strain to stress, and stress to loading, as follows. In uniaxial loading, as in the bar element, we need consider only the normal strain component, defined as

$$\varepsilon_x = \frac{\mathrm{d}u}{\mathrm{d}x} \tag{2.29}$$

which, when applied to Equation 2.25, gives

$$\varepsilon_x = \frac{u_2 - u_1}{L} \tag{2.30}$$

which shows that the spar element is a constant strain element. This is in accord with strength of materials theory: The element has constant cross-sectional area and is subjected to constant forces at the end points, so the strain does not vary along the length. The axial stress, by Hooke's law, is then

$$\sigma_x = E\varepsilon_x = E\frac{u_2 - u_1}{L} \tag{2.31}$$

and the associated axial force is

$$P = \sigma_x A = \frac{AE}{L}(u_2 - u_1) \tag{2.32}$$

Taking care to observe the correct algebraic sign convention, Equation 2.32 is now used to relate the applied nodal forces f_1 and f_2 to the nodal displacements u_1 and u_2. Observing that, if Equation 2.32 has a positive sign, the element is in tension and nodal force f_2 must be in the positive coordinate direction while nodal force f_1 must be equal and opposite for equilibrium; therefore,

$$f_1 = -\frac{AE}{L}(u_2 - u_1) \tag{2.33}$$

$$f_2 = \frac{AE}{L}(u_2 - u_1) \tag{2.34}$$

Equations 2.33 and 2.34 are expressed in matrix form as

$$\frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}\begin{Bmatrix} u_1 \\ u_2 \end{Bmatrix} = \begin{Bmatrix} f_1 \\ f_2 \end{Bmatrix} \tag{2.35}$$

Comparison of Equation 2.35 to Equation 2.4 shows that the stiffness matrix for the bar element is given by

$$[k_e] = \frac{AE}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{2.36}$$

As is the case with the linear spring, we observe that the element stiffness matrix for the bar element is symmetric, singular, and of order 2×2 in correspondence with two nodal displacements or *degrees of freedom*. It must be emphasized that the stiffness matrix given by Equation 2.36 is expressed in the *element coordinate system,* which in this case is one-dimensional. Application of this element formulation to analysis of two- and three-dimensional structures is considered in the next chapter.

EXAMPLE 2.4

Figure 2.7a depicts a tapered elastic bar subjected to an applied tensile load P at one end and attached to a fixed support at the other end. The cross-sectional area varies linearly from A_0 at the fixed support at $x = 0$ to $A_0/2$ at $x = L$. Calculate the displacement of the end of the bar (a) by modeling the bar as a single element having cross-sectional area equal to the area of the actual bar at its midpoint along the length, (b) using two bar elements of equal length and similarly evaluating the area at the midpoint of each, and (c) using integration to obtain the exact solution.

■ **Solution**

(a) For a single element, the cross-sectional area is $3A_0/4$ and the element "spring constant" is

$$k = \frac{AE}{L} = \frac{3A_0E}{4L}$$

and the element equations are

$$\frac{3A_0E}{4L}\begin{bmatrix} 1 & -1 \\ -1 & -1 \end{bmatrix}\begin{Bmatrix} U_1 \\ U_2 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ P \end{Bmatrix}$$

The element and nodal displacements are as shown in Figure 2.7b. Applying the constraint condition $U_1 = 0$, we find

$$U_2 = \frac{4PL}{3A_0E} = 1.333\frac{PL}{A_0E}$$

as the displacement at $x = L$.

(b) Two elements of equal length $L/2$ with associated nodal displacements are depicted in Figure 2.7c. For element 1, $A_1 = 7A_0/8$ so

$$k_1 = \frac{A_1E}{L_1} = \frac{7A_0E}{8(L/2)} = \frac{7A_0E}{4L}$$

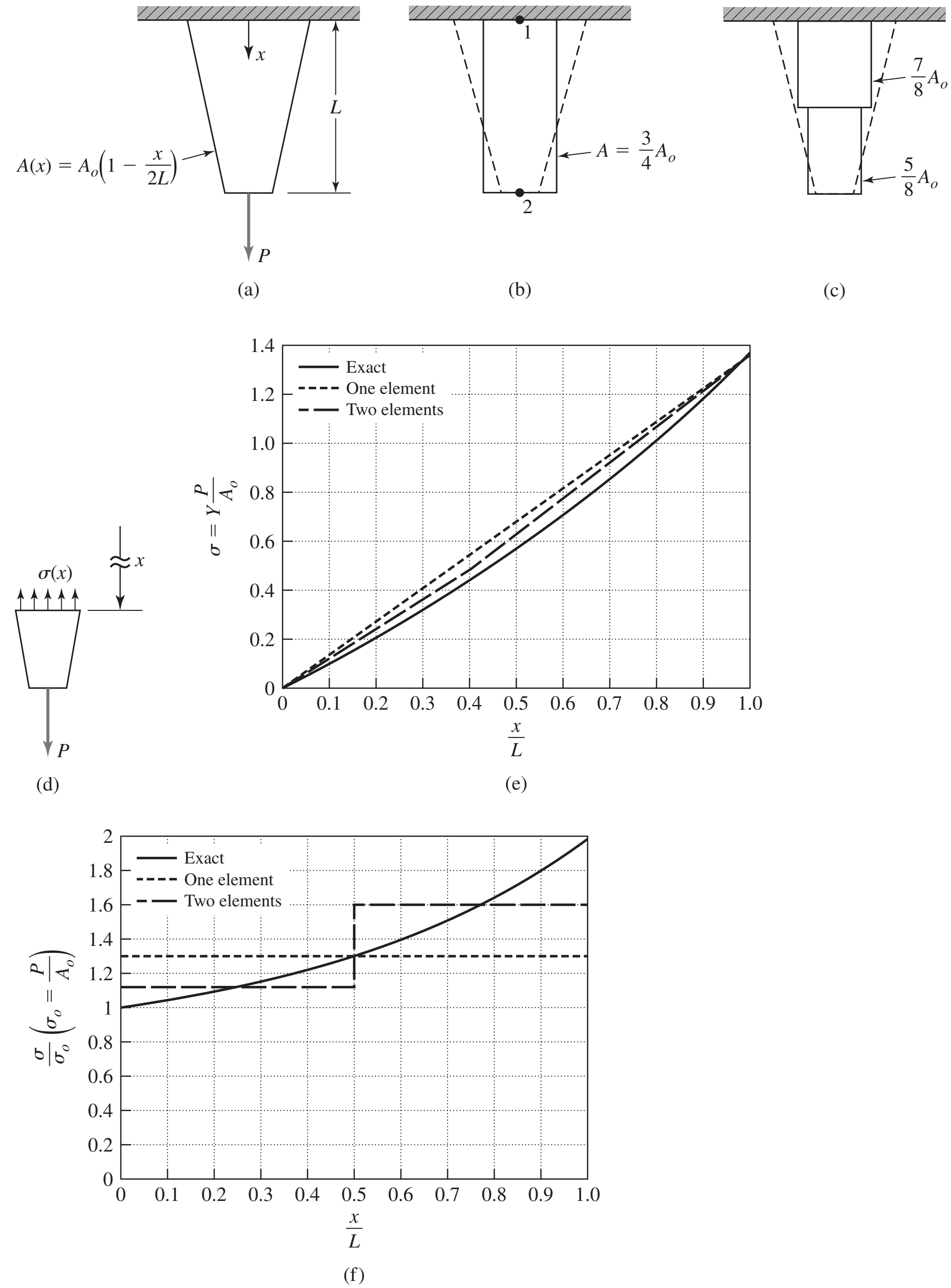


Figure 2.7
(a) Tapered axial bar, (b) one-element model, (c) two-element model, (d) free-body diagram for an exact solution, (e) displacement solutions, (f) stress solutions.

while for element 2, we have

$$A_1 = \frac{5A_0}{8} \quad \text{and} \quad k_2 = \frac{A_2 E}{L_2} = \frac{5A_0 E}{8(L/2)} = \frac{5A_0 E}{4L}$$

Since no load is applied at the center of the bar, the equilibrium equations for the system of two elements is

$$\begin{bmatrix} k_1 & -k_1 & 0 \\ -k_1 & k_1 + k_2 & -k_2 \\ 0 & -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ 0 \\ P \end{Bmatrix}$$

Applying the constraint condition $U_1 = 0$ results in

$$\begin{bmatrix} k_1 + k_2 & -k_2 \\ -k_2 & k_2 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \end{Bmatrix} = \begin{Bmatrix} 0 \\ P \end{Bmatrix}$$

Adding the two equations gives

$$U_2 = \frac{P}{k_1} = \frac{4PL}{7A_0 E}$$

and substituting this result into the first equation results in

$$U_3 = \frac{k_1 + k_2}{k_2} = \frac{48PL}{35A_0 E} = 1.371\frac{PL}{A_0 E}$$

(c) To obtain the exact solution, we refer to Figure 2.7d, which is a free-body diagram of a section of the bar between an arbitrary position x and the end $x = L$. For equilibrium,

$$\sigma_x A = P \quad \text{and since} \quad A = A(x) = A_0\left(1 - \frac{x}{2L}\right)$$

the axial stress variation along the length of the bar is described by

$$\sigma_x = \frac{P}{A_0\left(1 - \dfrac{x}{2L}\right)}$$

Therefore, the axial strain is

$$\varepsilon_x = \frac{\sigma_x}{E} = \frac{P}{EA_0\left(1 - \dfrac{x}{2L}\right)}$$

Since the bar is fixed at $x = 0$, the displacement at $x = L$ is given by

$$\delta = \int_0^L \varepsilon_x \, \mathrm{d}x = \frac{P}{EA_0}\int_0^L \frac{\mathrm{d}x}{\left(1 - \dfrac{x}{2L}\right)}$$

$$= \frac{2PL}{EA_0}[-\ln(2L - x)]\Big|_0^L = \frac{2PL}{EA_0}[\ln(2L) - \ln L] = \frac{2PL}{EA_0}\ln 2 = 1.386\frac{PL}{A_0 E}$$

Comparison of the results of parts b and c reveals that the two element solution exhibits an error of only about 1 percent in comparison to the exact solution from strength of materials theory. Figure 2.7e shows the displacement variation along the length for the three solutions. It is extremely important to note, however, that the computed axial stress for the finite element solutions varies significantly from that of the exact solution. The axial stress for the two-element solution is shown in Figure 2.7f, along with the calculated stress from the exact solution. Note particularly the discontinuity of calculated stress values for the two elements at the connecting node. This is typical of the derived, or secondary, variables, such as stress and strain, as computed in the finite element method. As more and more smaller elements are used in the model, the values of such discontinuities decrease, indicating solution convergence. In structural analyses, the finite element user is most often more interested in stresses than displacements, hence it is essential that convergence of the secondary variables be monitored.

2.4 STRAIN ENERGY, CASTIGLIANO'S FIRST THEOREM

When external forces are applied to a body, the mechanical work done by those forces is converted, in general, into a combination of kinetic and potential energies. In the case of an elastic body constrained to prevent motion, all the work is stored in the body as elastic potential energy, which is also commonly referred to as *strain energy*. Here, strain energy is denoted U_e and mechanical work W. From elementary statics, the mechanical work performed by a force $\vec{F}$ as its point of application moves along a path from position 1 to position 2 is defined as

$$W = \int_1^2 \vec{F} \cdot \mathrm{d}\vec{r} \tag{2.37}$$

where

$$\mathrm{d}\vec{r} = \mathrm{d}x\vec{i} + \mathrm{d}y\vec{j} + \mathrm{d}z\vec{k} \tag{2.38}$$

is a differential vector along the path of motion. In Cartesian coordinates, work is given by

$$W = \int_{x_1}^{x_2} F_x\,\mathrm{d}x + \int_{y_1}^{y_2} F_y\,\mathrm{d}y + \int_{z_1}^{z_2} F_z\,\mathrm{d}z \tag{2.39}$$

where F_x, F_y, and F_z are the Cartesian components of the force vector.

For linearly elastic deformations, deflection is directly proportional to applied force as, for example, depicted in Figure 2.8 for a linear spring. The slope of the force-deflection line is the spring constant such that $F = k\delta$. Therefore, the work required to deform such a spring by an arbitrary amount δ_0 from its

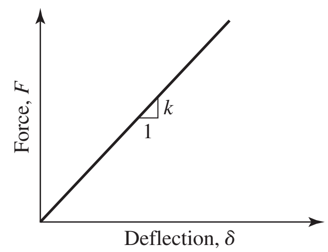


Figure 2.8 Force-deflection relation for a linear elastic spring.

free length is

$$W = \int_0^{\delta_0} F\,d\delta = \int_0^{\delta_0} k\delta\,d\delta = \frac{1}{2}k\delta_0^2 = U_e \tag{2.40}$$

and we observe that the work and resulting elastic potential energy are quadratic functions of displacement and have the units of force-length. This is a general result for linearly elastic systems, as will be seen in many examples throughout this text.

Utilizing Equation 2.28, the strain energy for an axially loaded elastic bar fixed at one end can immediately be written as

$$U_e = \frac{1}{2}k\delta^2 = \frac{1}{2}\frac{AE}{L}\delta^2 \tag{2.41}$$

However, for a more general purpose, this result is converted to a different form (applicable to a bar element only) as follows:

$$U_e = \frac{1}{2}k\delta^2 = \frac{1}{2}\frac{AE}{L}\left(\frac{PL}{AE}\right)^2 = \frac{1}{2}\left(\frac{P}{A}\right)\left(\frac{P}{AE}\right)AL = \frac{1}{2}\sigma\varepsilon V \tag{2.42}$$

where V is the total volume of deformed material and the quantity $\frac{1}{2}\sigma\varepsilon$ is *strain energy per unit volume,* also known as *strain energy density*. In Equation 2.42, stress and strain values are those corresponding to the *final* value of applied force. The factor $\frac{1}{2}$ arises from the linear relation between stress and strain as the load is applied from zero to the final value P. In general, for uniaxial loading, the strain energy per unit volume u_e is defined by

$$u_e = \int_0^{\varepsilon} \sigma\,d\varepsilon \tag{2.43}$$

which is extended to more general states of stress in subsequent chapters. We note that Equation 2.43 represents the area under the elastic stress-strain diagram.

Presently, we will use the work-strain energy relation to obtain the governing equations for the bar element using the following theorem.

Castigliano's First Theorem [1]

For an elastic system in equilibrium, the partial derivative of total strain energy with respect to deflection at a point is equal to the applied force in the direction of the deflection at that point.

Consider an elastic body subjected to N forces F_j for which the total strain energy is expressed as

$$U_e = W = \sum_{j=1}^{N} \int_0^{\delta_j} F_j \, \mathrm{d}\delta_j \tag{2.44}$$

where δ_j is the deflection at the point of application of force F_j in the direction of the line of action of the force. If all points of load application are fixed except one, say, i, and that point is made to deflect an infinitesimal amount $\Delta\delta_i$ by an incremental infinitesimal force ΔF_i, the change in strain energy is

$$\Delta U_e = \Delta W = F_i \Delta\delta_i + \int_0^{\Delta\delta_i} \Delta F_i \, \mathrm{d}\delta_i \tag{2.45}$$

where it is assumed that the original force F_i is constant during the infinitesimal change. The integral term in Equation 2.45 involves the product of infinitesimal quantities and can be neglected to obtain

$$\frac{\Delta U_e}{\Delta\delta_i} = F_i \tag{2.46}$$

which in the limit as $\Delta\delta_i$ approaches zero becomes

$$\frac{\partial U}{\partial \delta_i} = F_i \tag{2.47}$$

The first theorem of Castigliano is a powerful tool for finite element formulation, as is now illustrated for the bar element. Combining Equations 2.30, 2.31, and 2.43, total strain energy for the bar element is given by

$$U_e = \frac{1}{2}\sigma_x \varepsilon_x V = \frac{1}{2} E \left(\frac{u_2 - u_1}{L} \right)^2 AL \tag{2.48}$$

Applying Castigliano's theorem with respect to each displacement yields

$$\frac{\partial U_e}{\partial u_1} = \frac{AE}{L}(u_1 - u_2) = f_1 \tag{2.49}$$

$$\frac{\partial U_e}{\partial u_2} = \frac{AE}{L}(u_2 - u_1) = f_2 \tag{2.50}$$

which are observed to be identical to Equations 2.33 and 2.34.

The first theorem of Castigliano is also applicable to rotational displacements. In the case of rotation, the partial derivative of strain energy with respect to a rotational displacement is equal to the moment/torque applied at the point of concern in the sense of the rotation. The following example illustrates the application in terms of a simple torsional member.

EXAMPLE 2.5

A solid circular shaft of radius R and length L is subjected to constant torque T. The shaft is fixed at one end, as shown in Figure 2.9. Formulate the elastic strain energy in terms of the angle of twist θ at $x = L$ and show that Castigliano's first theorem gives the correct expression for the applied torque.

■ Solution

From strength of materials theory, the shear stress at any cross section along the length of the member is given by

$$\tau = \frac{Tr}{J}$$

where r is radial distance from the axis of the member and J is polar moment of inertia of the cross section. For elastic behavior, we have

$$\gamma = \frac{\tau}{G} = \frac{Tr}{JG}$$

where G is the shear modulus of the material, and the strain energy is then

$$U_e = \frac{1}{2}\int_V \tau\gamma \, \mathrm{d}V = \frac{1}{2}\int_0^L \left[\int_A \left(\frac{Tr}{J}\right)\left(\frac{Tr}{JG}\right) \mathrm{d}A\right] \mathrm{d}x$$

$$= \frac{T^2}{2J^2G}\int_0^L\int_A r^2 \, \mathrm{d}A \, \mathrm{d}x = \frac{T^2L}{2JG}$$

where we have used the definition of the polar moment of inertia

$$J = \int_A r^2 \, \mathrm{d}A$$

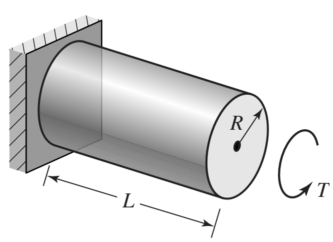


Figure 2.9 Example 2.5: Circular cylinder subjected to torsion.

Again invoking the strength of materials results, the angle of twist at the end of the member is known to be

$$\theta = \frac{TL}{JG}$$

so the strain energy can be written as

$$U_e = \frac{1}{2}\frac{L}{JG}\left(\frac{JG\theta}{L}\right)^2 = \frac{JG}{2L}\theta^2$$

Per Castangliano's first theorem,

$$\frac{\partial U_e}{\partial \theta} = T = \frac{JG\theta}{L}$$

which is exactly the relation shown by strength of materials theory. The reader may think that we used circular reasoning in this example, since we utilized many previously known results. However, the formulation of strain energy must be based on known stress and strain relationships, and the application of Castigliano's theorem is, indeed, a different concept.

For linearly elastic systems, formulation of the strain energy function in terms of displacements is relatively straightforward. As stated previously, the strain energy for an elastic system is a quadratic function of displacements. The quadratic nature is simplistically explained by the facts that, in elastic deformation, stress is proportional to force (or moment or torque), stress is proportional to strain, and strain is proportional to displacement (or rotation). And, since the elastic strain energy is equal to the mechanical work expended, a quadratic function results. Therefore, application of Castigliano's first theorem results in linear algebraic equations that relate displacements to applied forces. This statement follows from the fact that a derivative of a quadratic term is linear. The coefficients of the displacements in the resulting equations are the components of the stiffness matrix of the system for which the strain energy function is written. Such an energy-based approach is the simplest, most-straightforward method for establishing the stiffness matrix of many structural finite elements.

EXAMPLE 2.6

(a) Apply Castigliano's first theorem to the system of four spring elements depicted in Figure 2.10 to obtain the system stiffness matrix. The vertical members at nodes 2 and 3 are to be considered rigid.
(b) Solve for the displacements and the reaction force at node 1 if

$$k_1 = 4 \text{ N/mm} \qquad k_2 = 6 \text{ N/mm} \qquad k_3 = 3 \text{ N/mm}$$

$$F_2 = -30 \text{ N} \qquad F_3 = 0 \qquad F_4 = 50 \text{ N}$$

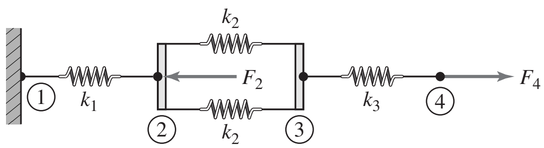


Figure 2.10 Example 2.6: Four spring elements.

■ Solution

(a) The total strain energy of the system of four springs is expressed in terms of the nodal displacements and spring constants as

$$U_e = \frac{1}{2}k_1(U_2 - U_1)^2 + 2\left[\frac{1}{2}k_2(U_3 - U_2)^2\right] + \frac{1}{2}k_3(U_4 - U_3)^2$$

Applying Castigliano's theorem, using each nodal displacement in turn,

$$\frac{\partial U_e}{\partial U_1} = F_1 = k_1(U_2 - U_1)(-1) = k_1(U_1 - U_2)$$

$$\frac{\partial U_e}{\partial U_2} = F_2 = k_1(U_2 - U_1) + 2k_2(U_3 - U_2)(-1) = -k_1U_1 + (k_1 + 2k_2)U_2 - 2k_2U_3$$

$$\frac{\partial U_e}{\partial U_3} = F_3 = 2k_2(U_3 - U_2) + k_3(U_4 - U_3)(-1) = -2k_2U_2 + (2k_2 + k_3)U_3 - k_3U_4$$

$$\frac{\partial U_e}{\partial U_4} = F_4 = k_3(U_4 - U_3) = -k_3U_3 + k_3U_4$$

which can be written in matrix form as

$$\begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & k_1 + 2k_2 & -2k_2 & 0 \\ 0 & -2k_2 & 2k_2 + k_3 & -k_3 \\ 0 & 0 & -k_3 & k_3 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix}$$

and the system stiffness matrix is thus obtained via Castigliano's theorem.

(b) Substituting the specified numerical values, the system equations become

$$\begin{bmatrix} 4 & -4 & 0 & 0 \\ -4 & 16 & -12 & 0 \\ 0 & -12 & 15 & -3 \\ 0 & 0 & -3 & 3 \end{bmatrix} \begin{Bmatrix} 0 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ -30 \\ 0 \\ 50 \end{Bmatrix}$$

Eliminating the constraint equation, the active displacements are governed by

$$\begin{bmatrix} 16 & -12 & 0 \\ -12 & 15 & -3 \\ 0 & -3 & 3 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} -30 \\ 0 \\ 50 \end{Bmatrix}$$

which we solve by manipulating the equations to convert the coefficient matrix (the stiffness matrix) to upper-triangular form; that is, all terms below the main diagonal become zero.

Step 1. Multiply the first equation (row) by 12, multiply the second equation (row) by 16, add the two and replace the second equation with the resulting equation to obtain

$$\begin{bmatrix} 16 & -12 & 0 \\ 0 & 96 & -48 \\ 0 & -3 & 3 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} -30 \\ -360 \\ 50 \end{Bmatrix}$$

Step 2. Multiply the third equation by 32, add it to the second equation, and replace the third equation with the result. This gives the triangularized form desired:

$$\begin{bmatrix} 16 & -12 & 0 \\ 0 & 96 & -48 \\ 0 & 0 & 48 \end{bmatrix} \begin{Bmatrix} U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} -30 \\ -360 \\ 1240 \end{Bmatrix}$$

In this form, the equations can now be solved from the "bottom to the top," and it will be found that, at each step, there is only one unknown. In this case, the sequence is

$$U_4 = \frac{1240}{48} = 25.83 \text{ mm}$$

$$U_3 = \frac{1}{96}[-360 + 48(25.83)] = 9.17 \text{ mm}$$

$$U_2 = \frac{1}{16}[-30 + 12(9.17)] = 5.0 \text{ mm}$$

The reaction force at node 1 is obtained from the constraint equation

$$F_1 = -4U_2 = -4(5.0) = -20 \text{ N}$$

and we observe system equilibrium since the external forces sum to zero as required.

2.5 MINIMUM POTENTIAL ENERGY

The first theorem of Castigliano is but a forerunner to the general principle of *minimum potential energy*. There are many ways to state this principle, and it has been proven rigorously [2]. Here, we state the principle without proof but expect the reader to compare the results with the first theorem of Castigliano. The principle of minimum potential energy is stated as follows:

Of all displacement states of a body or structure, subjected to external loading, that satisfy the geometric boundary conditions (imposed displacements), the displacement state that also satisfies the equilibrium equations is such that the total *potential energy is a minimum for stable equilibrium.*

We emphasize that the *total* potential energy must be considered in application of this principle. The total potential energy includes the stored elastic potential energy (the strain energy) as well as the potential energy of applied loads. As is customary, we use the symbol Π for total potential energy and divide the total potential energy into two parts, that portion associated with strain energy U_e and the portion associated with external forces U_F. The total potential energy is

$$\Pi = U_e + U_F \tag{2.51}$$

where it is to be noted that the term external *forces* also includes moments and torques.

In this text, we will deal only with elastic systems subjected to *conservative* forces. A *conservative force* is defined as one that does mechanical work independent of the path of motion and such that the work is reversible or recoverable. The most common example of a *nonconservative* force is the force of sliding friction. As the friction force always acts to oppose motion, the work done by friction forces is always negative and results in energy loss. This loss shows itself physically as generated heat. On the other hand, the mechanical work done by a conservative force, Equation 2.37, is reversed, and therefore recovered, if the force is released. Therefore, the mechanical work of a conservative force is considered to be a loss in potential energy; that is,

$$U_F = -W \tag{2.52}$$

where W is the mechanical work defined by the scalar product integral of Equation 2.37. The total potential energy is then given by

$$\Pi = U_e - W \tag{2.53}$$

As we show in the following examples and applications to solid mechanics in Chapter 9, the strain energy term U_e is a quadratic function of system displacements and the work term W is a linear function of displacements. Rigorously, the minimization of total potential energy is a problem in the *calculus of variations* [5]. We do not suppose that the intended audience of this text is familiar with the calculus of variations. Rather, we simply impose the minimization principle of calculus of multiple variable functions. If we have a total potential energy expression that is a function of, say, N displacements U_i, $i = 1, \ldots, N$; that is,

$$\Pi = \Pi(U_1, U_2, \ldots, U_N) \tag{2.54}$$

then the total potential energy will be minimized if

$$\frac{\partial \Pi}{\partial U_i} = 0 \qquad i = 1, \ldots, N \tag{2.55}$$

Equation 2.55 will be shown to represent N algebraic equations, which form the finite element approximation to the solution of the differential equation(s) governing the response of a structural system.

EXAMPLE 2.7

Repeat the solution to Example 2.6 using the principle of minimum potential energy.

■ **Solution**

Per the previous example solution, the elastic strain energy is

$$U_e = \frac{1}{2}k_1(U_2 - U_1)^2 + 2\left[\frac{1}{2}k_2(U_3 - U_2)^2\right] + \frac{1}{2}k_3(U_4 - U_3)^2$$

and the potential energy of applied forces is

$$U_F = -W = -F_1U_1 - F_2U_2 - F_3U_3 - F_4U_4$$

Hence, the total potential energy is expressed as

$$\Pi = \frac{1}{2}k_1(U_2 - U_1)^2 + 2\left[\frac{1}{2}k_2(U_3 - U_2)^2\right] + \frac{1}{2}k_3(U_4 - U_3)^2 - F_1U_1 - F_2U_2 - F_3U_3 - F_4U_4$$

In this example, the principle of minimum potential energy requires that

$$\frac{\partial \Pi}{\partial U_i} = 0 \qquad i = 1, 4$$

giving in sequence $i = 1, 4$, the algebraic equations

$$\frac{\partial \Pi}{\partial U_1} = k_1(U_2 - U_1)(-1) - F_1 = k_1(U_1 - U_2) - F_1 = 0$$

$$\frac{\partial \Pi}{\partial U_2} = k_1(U_2 - U_1) + 2k_2(U_3 - U_2)(-1) - F_2$$

$$= -k_1U_1 + (k_1 + 2k_2)U_2 - 2k_2U_3 - F_2 = 0$$

$$\frac{\partial \Pi}{\partial U_3} = 2k_2(U_3 - U_2) + k_3(U_4 - U_3)(-1) - F_3$$

$$= -2k_2U_2 + (2k_2 + k_3)U_3 - k_3U_4 - F_3 = 0$$

$$\frac{\partial \Pi}{\partial U_4} = k_3(U_4 - U_3) - F_4 = -k_3U_3 + k_3U_4 - F_4 = 0$$

which, when written in matrix form, are

$$\begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & k_1 + 2k_2 & -2k_2 & 0 \\ 0 & -2k_2 & 2k_2 + k_3 & -k_3 \\ 0 & 0 & -k_3 & k_3 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} = \begin{Bmatrix} F_1 \\ F_2 \\ F_3 \\ F_4 \end{Bmatrix}$$

and can be seen to be identical to the previous result. Consequently, we do not resolve the system numerically, as the results are known.

We now reexamine the energy equation of the Example 2.7 to develop a more-general form, which will be of significant value in more complicated systems to be discussed in later chapters. The system or global displacement vector is

$$\{U\} = \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} \tag{2.56}$$

and, as derived, the global stiffness matrix is

$$[K] = \begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & k_1 + 2k_2 & -2k_2 & 0 \\ 0 & -2k_2 & 2k_2 + k_3 & -k_3 \\ 0 & 0 & -k_3 & k_3 \end{bmatrix} \tag{2.57}$$

If we form the matrix triple product

$$\frac{1}{2}\{U\}^T[K]\{U\} = \frac{1}{2}[\,U_1 \quad U_2 \quad U_3 \quad U_4\,]$$
$$\times \begin{bmatrix} k_1 & -k_1 & 0 & 0 \\ -k_1 & k_1 + 2k_2 & -2k_2 & 0 \\ 0 & -2k_2 & 2k_2 + k_3 & -k_3 \\ 0 & 0 & -k_3 & k_3 \end{bmatrix} \begin{Bmatrix} U_1 \\ U_2 \\ U_3 \\ U_4 \end{Bmatrix} \tag{2.58}$$

and carry out the matrix operations, we find that the expression is identical to the strain energy of the system. As will be shown, the matrix triple product of Equation 2.58 represents the strain energy of any elastic system. If the strain energy can be expressed in the form of this triple product, the stiffness matrix will have been obtained, since the displacements are readily identifiable.

2.6 SUMMARY

Two linear mechanical elements, the idealized elastic spring and an elastic tension-compression member (bar) have been used to introduce the basic concepts involved in formulating the equations governing a finite element. The element equations are obtained by both a straightforward equilibrium approach and a strain energy method using the first theorem of Castigliano. The principle of minimum potential also is introduced. The next chapter shows how the one-dimensional bar element can be used to demonstrate the finite element model assembly procedures in the context of some simple two- and three-dimensional structures.

REFERENCES

1. Budynas, R. *Advanced Strength and Applied Stress Analysis.* 2d ed. New York: McGraw-Hill, 1998.
2. Love, A. E. H. *A Treatise on the Mathematical Theory of Elasticity.* New York: Dover Publications, 1944.

3. Beer, F. P., E. R. Johnston, and J. T. DeWolf. *Mechanics of Materials.* 3d ed. New York: McGraw-Hill, 2002.

4. Shigley, J., and R. Mischke. *Mechanical Engineering Design*. New York: McGraw-Hill, 2001.

5. Forray, M. J. *Variational Calculus in Science and Engineering*. New York: McGraw-Hill, 1968.

PROBLEMS

2.1–2.3 For each assembly of springs shown in the accompanying figures (Figures P2.1–P2.3), determine the global stiffness matrix using the system assembly procedure of Section 2.2.

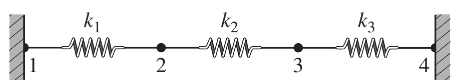


Figure P2.1

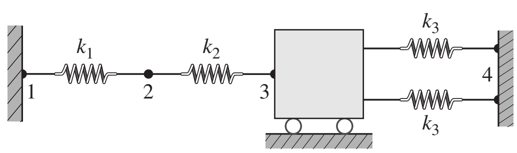


Figure P2.2

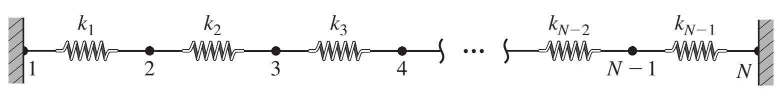


Figure P2.3

2.4 For the spring assembly of Figure P2.4, determine force F_3 required to displace node 2 an amount $\delta = 0.75$ in. to the right. Also compute displacement of node 3. Given

$$k_1 = 50 \text{ lb./in.} \quad \text{and} \quad k_2 = 25 \text{ lb./in.}$$

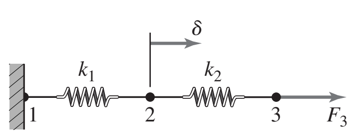
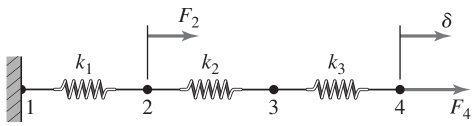


Figure P2.4

2.5 In the spring assembly of Figure P2.5, forces F_2 and F_4 are to be applied such that the resultant force in element 2 is zero and node 4 displaces an amount

$\delta = 1$ in. Determine (a) the required values of forces F_2 and F_4, (b) displacement of node 2, and (c) the reaction force at node 1.



$k_1 = k_3 = 30$ lb./in. $\quad k_2 = 40$ lb./in.

Figure P2.5

2.6 Verify the global stiffness matrix of Example 2.3 using (a) direct assembly and (b) Castigliano's first theorem.

2.7 Two trolleys are connected by the arrangement of springs shown in Figure P2.7. (a) Determine the complete set of equilibrium equations for the system in the form $[K]\{U\} = \{F\}$. (b) If $k = 50$ lb./in., $F_1 = 20$ lb., and $F_2 = 15$ lb., compute the displacement of each trolley and the force in each spring.

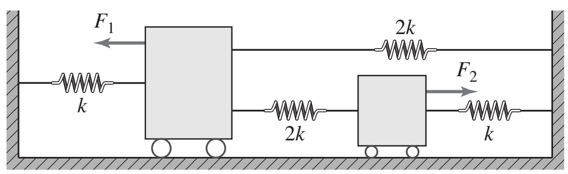


Figure P2.7

2.8 Use Castigliano's first theorem to obtain the matrix equilibrium equations for the system of springs shown in Figure P2.8.

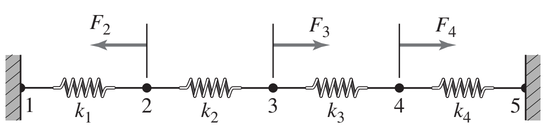
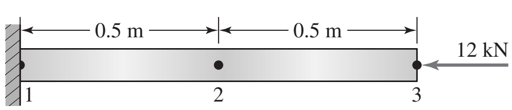


Figure P2.8

2.9 In Problem 2.8, let $k_1 = k_2 = k_3 = k_4 = 10$ N/mm, $F_2 = 20$ N, $F_3 = 25$ N, $F_4 = 40$ N and solve for (a) the nodal displacements, (b) the reaction forces at nodes 1 and 5, and (c) the force in each spring.

2.10 A steel rod subjected to compression is modeled by two bar elements, as shown in Figure P2.10. Determine the nodal displacements and the axial stress in each element. What other concerns should be examined?



$E = 207$ GPa $\quad A = 500$ mm^2

Figure P2.10

2.11 Figure P2.11 depicts an assembly of two bar elements made of different materials. Determine the nodal displacements, element stresses, and the reaction force.

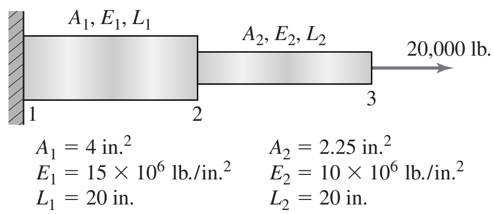


Figure P2.11

2.12 Obtain a four-element solution for the tapered bar of Example 2.4. Plot element stresses versus the exact solution. Use the following numerical values:

$$E = 10 \times 10^6 \text{ lb./in.}^2 \qquad A_0 = 4 \text{ in.}^2 \qquad L = 20 \text{ in.} \qquad P = 4000 \text{ lb.}$$

2.13 A weight W is suspended in a vertical plane by a linear spring having spring constant k. Show that the equilibrium position corresponds to minimum total potential energy.

2.14 For a bar element, it is proposed to discretize the displacement function as

$$u(x) = N_1(x)u_1 + N_2(x)u_2$$

with interpolation functions

$$N_1(x) = \cos\frac{\pi x}{2L}$$

$$N_2(x) = \sin\frac{\pi x}{2L}$$

Are these valid interpolation functions? (Hint: Consider strain and stress variations.)

2.15 The torsional element shown in Figure P2.15 has a solid circular cross section and behaves elastically. The nodal displacements are rotations θ_1 and θ_2 and the associated nodal loads are applied torques T_1 and T_2. Use the potential energy principle to derive the element equations in matrix form.

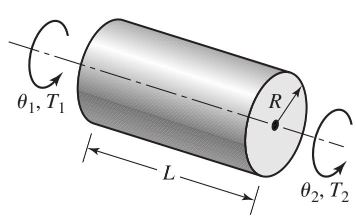


Figure P2.15

CHAPTER 3

Truss Structures: The Direct Stiffness Method

3.1 INTRODUCTION

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

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

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

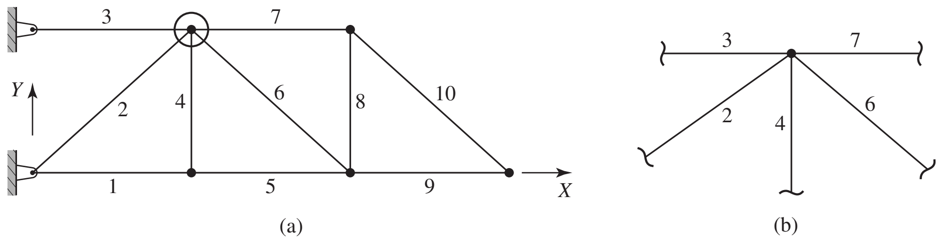


Figure 3.1
(a) Two-dimensional truss composed of ten elements. (b) Truss joint connecting five elements.

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

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

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

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

3.2 NODAL EQUILIBRIUM EQUATIONS

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

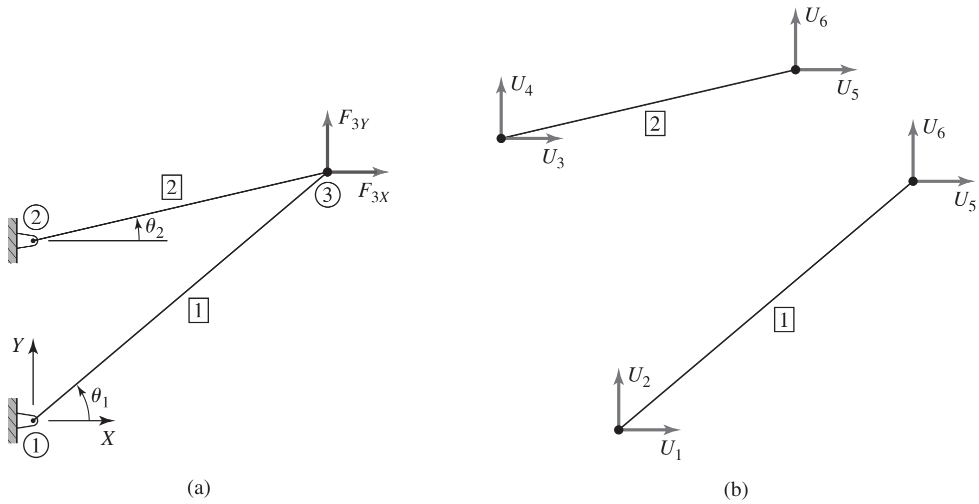


Figure 3.2
(a) A two-element truss with node and element numbers. (b) Global displacement notation.

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

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

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

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

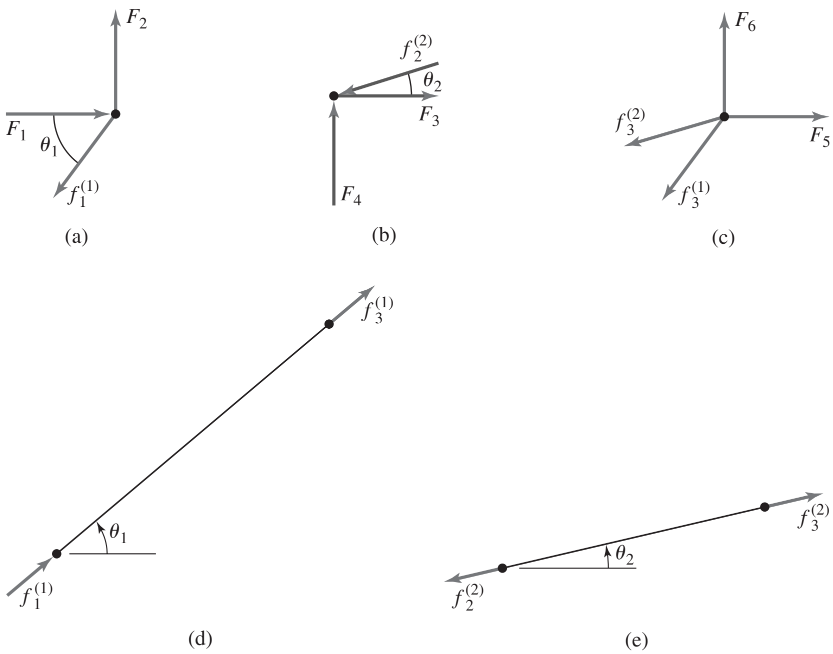


Figure 3.3
(a)–(c) Nodal free-body diagrams. (d) and (e) Element free-body diagrams.

and for node 2,

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

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

while for node 3,

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

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

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

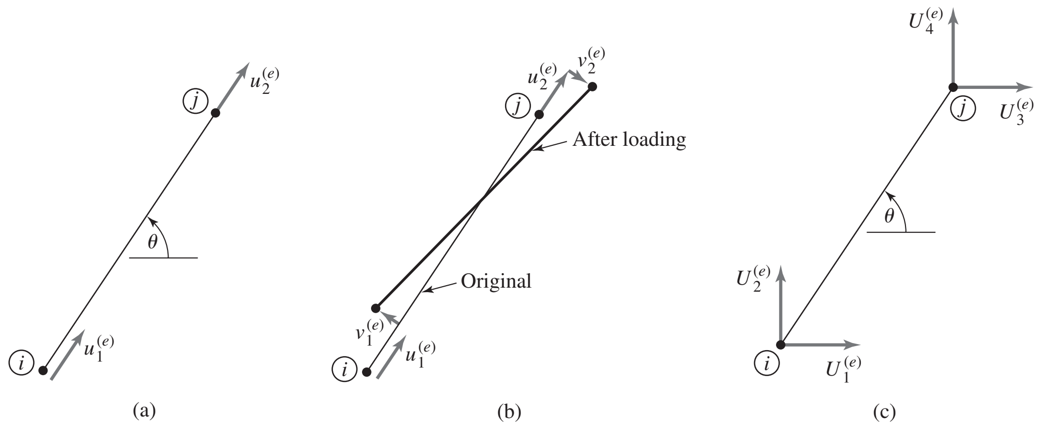


Figure 3.4
(a) Bar element at orientation θ. (b) General displacements of a bar element. (c) Bar element global displacements.

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

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

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

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

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

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

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

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

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

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

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

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

The net axial force acting on the element is then

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

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

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

and similarly for element 2 (Figure 3.3e):

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

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

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

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

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

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

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

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

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

Equations 3.9 through 3.14 are equivalent to the matrix form

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

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

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

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

3.3 ELEMENT TRANSFORMATION

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

Recalling the bar element equations expressed in the element frame as

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

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

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

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

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

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

which can be written in matrix form as

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

where

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

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

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

or

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

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

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

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

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

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

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

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

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

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

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

3.3.1 Direction Cosines

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

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

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

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

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

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

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

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

3.4 DIRECT ASSEMBLY OF GLOBAL STIFFNESS MATRIX

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

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

for element 1 and

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

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

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

while for element 2

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

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

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

Table 3.1 Nodal Displacement Correspondence Table

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

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

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

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

EXAMPLE 3.1

For the truss shown in Figure 3.2, $\theta_1 = \pi/4$, $\theta_2 = 0$, and the element properties are such that $k_1 = A_1 E_1/L_1$, $k_2 = A_2 E_2/L_2$. Transform the element stiffness matrix of each element into the global reference frame and assemble the global stiffness matrix.

■ Solution

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

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

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

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

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

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

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

The complete global stiffness matrix is then

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

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

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

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

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

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

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

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

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

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

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

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

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

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

Table 3.2 Element-Node Connectivity Table for Figure 3.2

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

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

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

3.5 BOUNDARY CONDITIONS, CONSTRAINT FORCES

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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

3.6 ELEMENT STRAIN AND STRESS

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

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

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

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

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

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

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

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

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

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

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

EXAMPLE 3.2

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

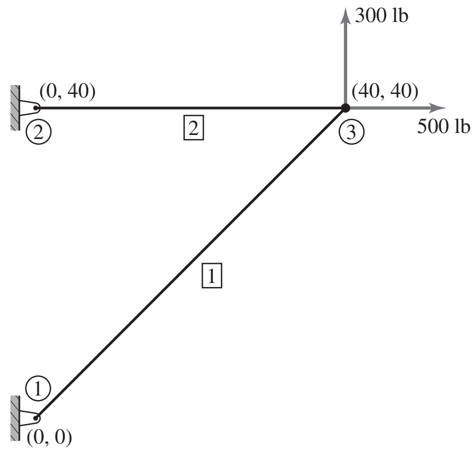


Figure 3.5 Two-element truss with external loading.

■ Solution

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

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

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

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

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

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

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

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

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

Simultaneous solution gives the displacements as

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

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

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

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

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

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

Element stress is computed using Equation 3.52:

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

Using the element displacements just computed, we have

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

and the positive results indicate tensile stress.

The element nodal forces via Equation 3.23 are

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

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

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

For element 2, the same procedure in sequence gives

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

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

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

also indicating tension.

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

3.7 COMPREHENSIVE EXAMPLE

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

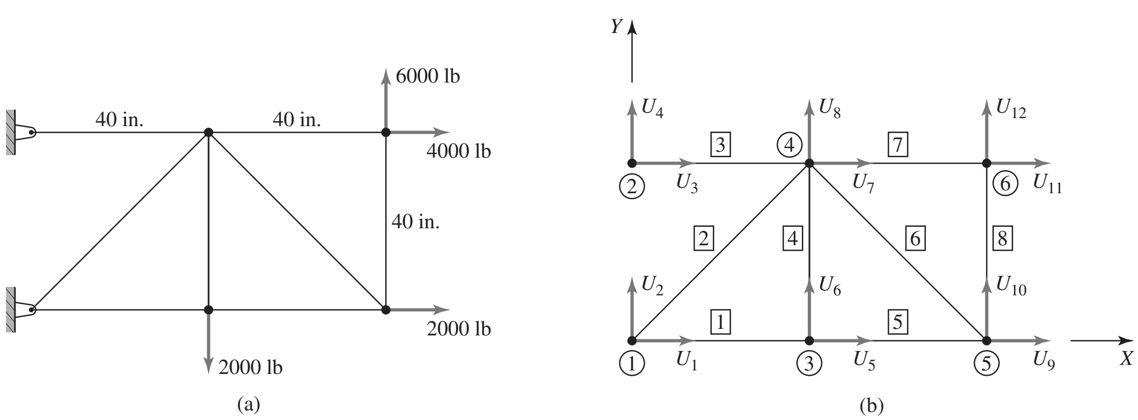


Figure 3.6
(a) For each element, $A = 1.5$ in.^{2}, $E = 10 \times 10^6$ psi. (b) Node, element, and global displacement notation.

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

Step 2. Compute individual element stiffness values:

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

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

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

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

we obtain

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

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

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

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

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

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

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

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

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

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

Table 3.3 Connectivity and Displacement Relations

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

Table 3.4 Element-Node Connectivity

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

$$L^{(5)} = [5 \quad 6 \quad 9 \quad 10]$$
$$L^{(6)} = [9 \quad 10 \quad 7 \quad 8]$$
$$L^{(7)} = [7 \quad 8 \quad 11 \quad 12]$$
$$L^{(8)} = [9 \quad 10 \quad 11 \quad 12]$$

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

$$K_{11} = k_{11}^{(1)} + k_{11}^{(2)} = (3.75 + 2.65/2)10^5$$
$$K_{12} = k_{12}^{(1)} + k_{12}^{(2)} = (0 + 2.65/2)10^5$$
$$K_{13} = K_{14} = 0$$
$$K_{15} = k_{13}^{(1)} = -3.75(10^5)$$
$$K_{16} = k_{14}^{(1)} = 0$$
$$K_{17} = k_{13}^{(2)} = -(2.65/2)10^5$$

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

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

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

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

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

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

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

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

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

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

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

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

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

and, on substitution of the computed displacements, yield

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

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

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

For element 2, for example, we have

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

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

The axial strain in element 2 is then

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

and corresponding axial stress is

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

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

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

Table 3.5 Results for the Eight Elements

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

3.8 THREE-DIMENSIONAL TRUSSES

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

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

or

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

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

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

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

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

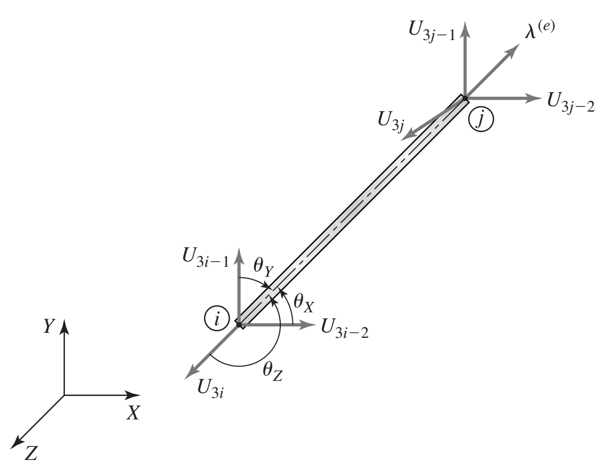


Figure 3.7 Bar element in a 3-D global coordinate system.

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

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

$$= [R]\{U^{(e)}\} \tag{3.57}$$

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

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

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

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

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

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

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

EXAMPLE 3.3

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

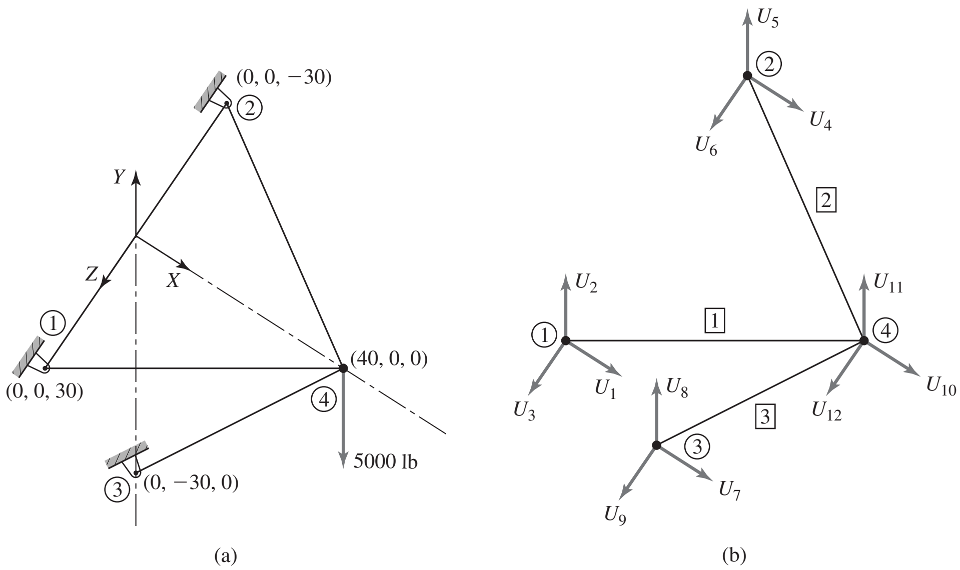


Figure 3.8
(a) A three-element, 3-D truss. (b) Numbering scheme.

■ Solution

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

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

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

Element 1

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

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

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

Table 3.6 Element-to-Global Displacement Correspondence

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

Element 2

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

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

Element 3

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

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

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

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

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

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

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

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

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

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

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

and simultaneous solution yields

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

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

$$U_{12} = 0$$

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

3.9 SUMMARY

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

REFERENCES

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

PROBLEMS

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

a. Using only static force equilibrium equations, solve for the force in each member as well as the reaction force components.
b. Assuming each member has axial stiffness $k = 52000$ lb/in., compute the axial deflection of each member.
c. Using the results of part b, calculate the X and Y displacements of node 3.

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

3.3 Verify Equation 3.28 by direct multiplication of the matrices.

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

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

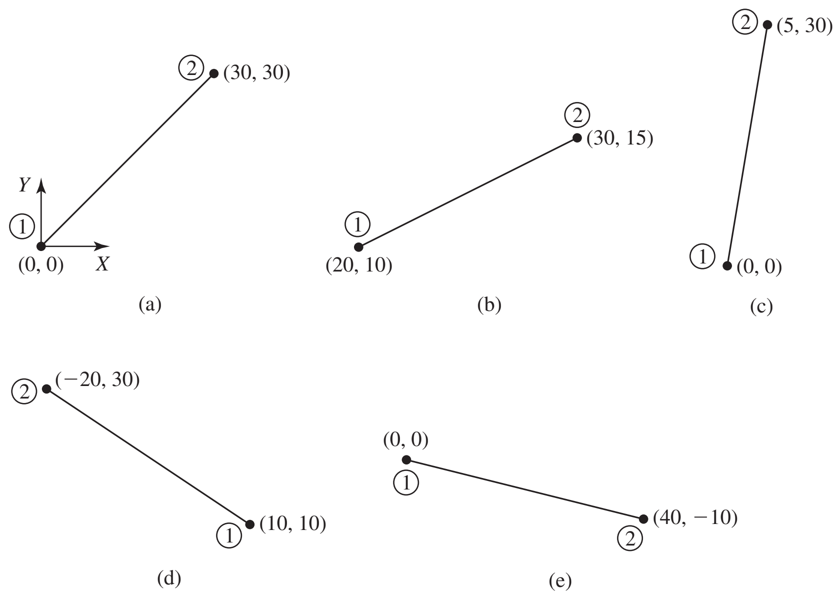


Figure P3.5

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

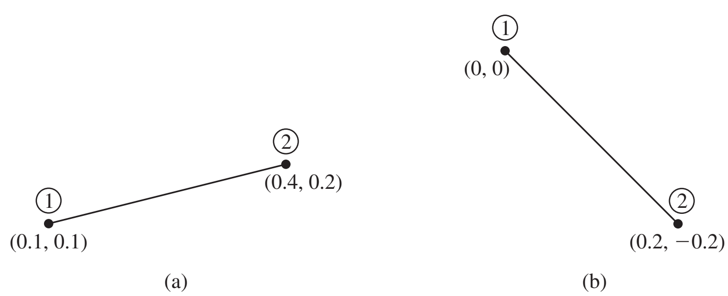


Figure P3.6

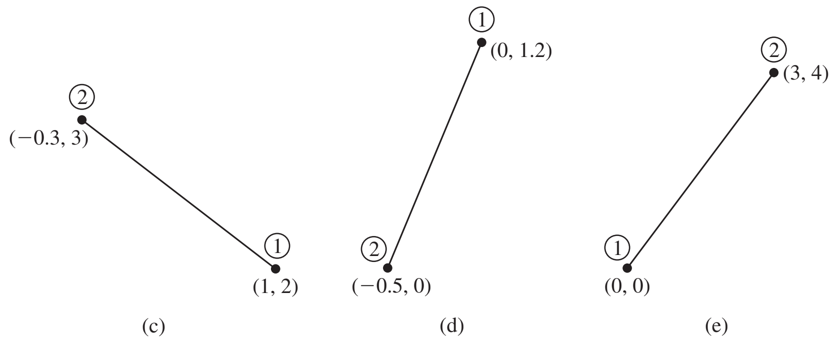


Figure P3.6 (*Continued*)

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

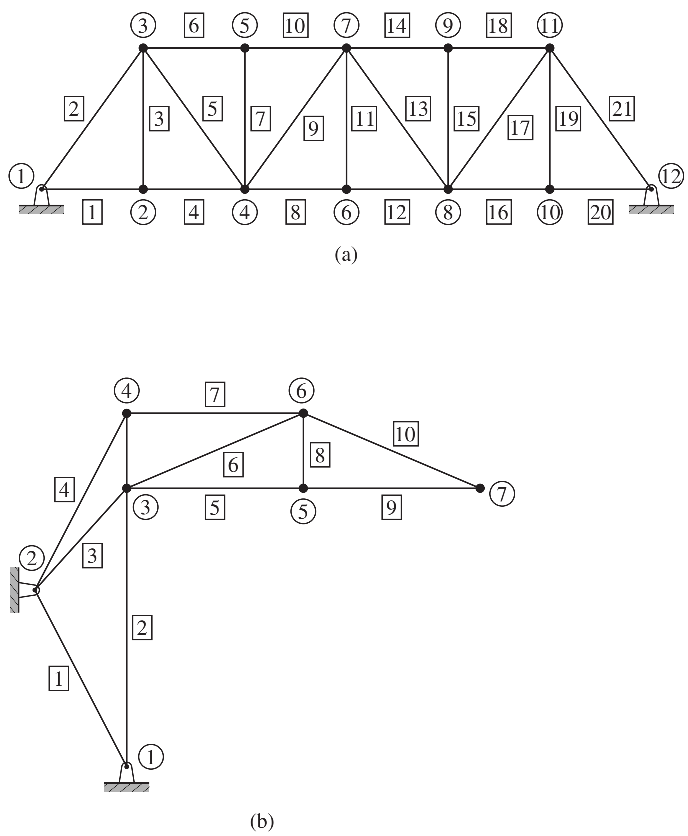
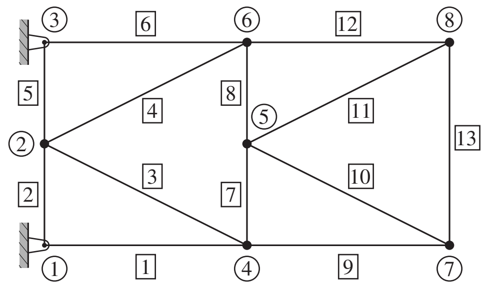
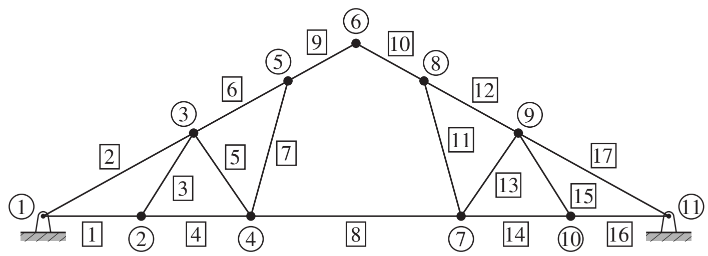


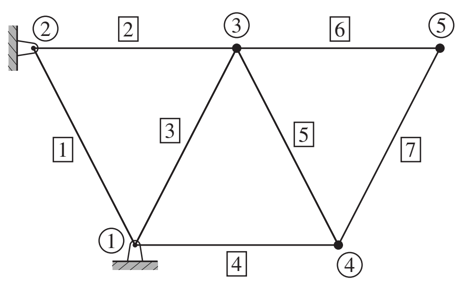
Figure P3.7



(c)



(d)



(e)

Figure P3.7 (*Continued*)

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

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

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

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

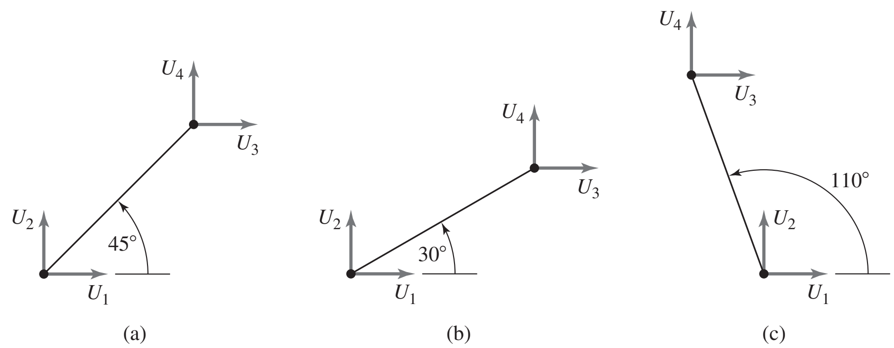


Figure P3.9

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

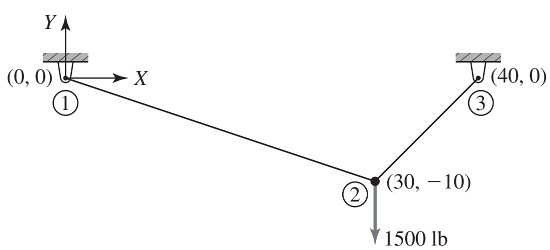


Figure P3.10

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

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

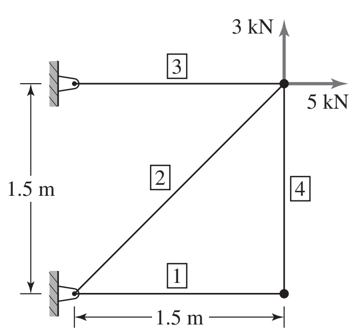
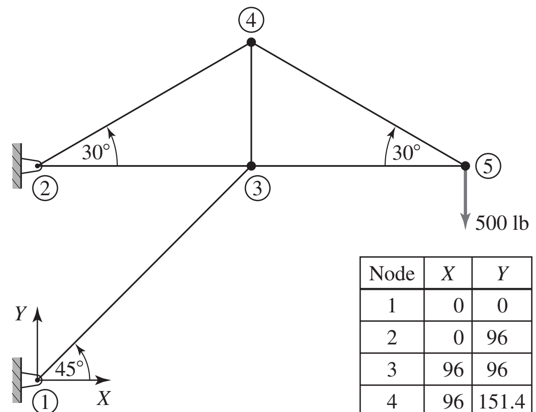


Figure P3.11

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

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

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



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

(inches)

Figure P3.13

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

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

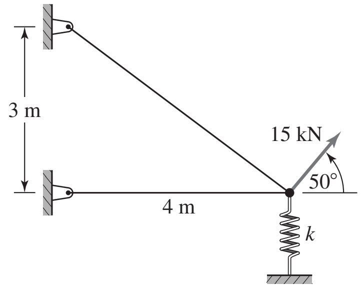


Figure P3.14

3.15 Repeat Problem 3.14 if the spring is removed.

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

3.17 Given the following system of algebraic equations

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

and the specified conditions

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

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

3.18 Given the system

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

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

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

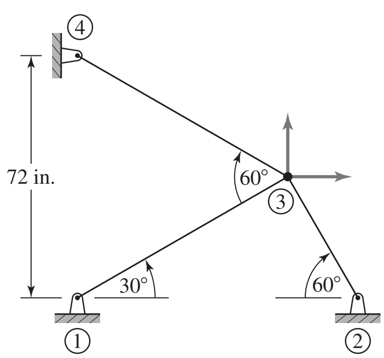


Figure P3.19

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

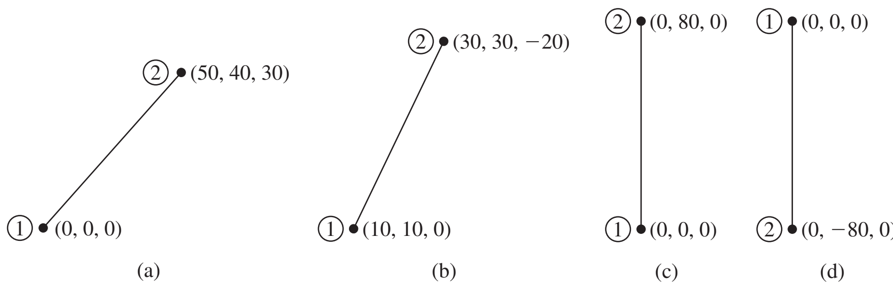


Figure P3.20

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

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

$$\sigma = E\varepsilon = E\begin{bmatrix} \dfrac{\mathrm{d}N_1}{\mathrm{d}x} & \dfrac{\mathrm{d}N_2}{\mathrm{d}x} \end{bmatrix}\begin{Bmatrix} u_1^{(e)} \\ u_2^{(e)} \end{Bmatrix} = E\begin{bmatrix} -\dfrac{1}{L} & \dfrac{1}{L} \end{bmatrix}[R]\{U^{(e)}\}$$

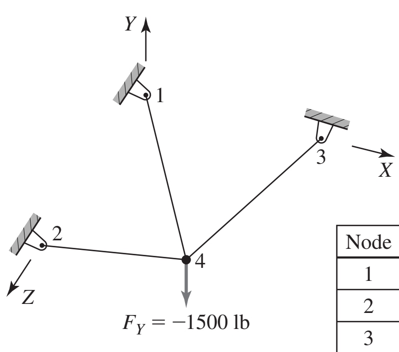
and note that the expression is the same as for the 2-D case.

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

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

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

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



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

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