GLOBAL EDITION

Finite Element Analysis

Theory and Application with ANSYS

FOURTH EDITION

Saeed Moaveni

ALWAYS LEARNING

PEARSON

FINITE ELEMENT ANALYSIS

FINITE ELEMENT ANALYSIS

Theory and Application with ANSYS

Fourth Edition

Global Edition

Saeed Moaveni

Minnesota State University, Mankato

PEARSON

Boston Columbus Indianapolis New York San Francisco Hoboken
Amsterdam Cape Town Dubai London Madrid Milan Munich Paris Montreal Toronto
Delhi Mexico City São Paulo Sydney Hong Kong Seoul Singapore Taipei Tokyo

Vice President and Editorial Director, ECS: *Marcia J. Horton*
Acquisitions Editor: *Norrin Dias*
Marketing Assistant: *Jon Bryant*
Senior Managing Editor: *Scott Disanno*
Production Program Manager: *Clare Romeo*
Production Project Manager: *Jennifer Sargunar*
Director of Operations: *Nick Sklitsis*
Operations Specialist: *Linda Sager*
Cover Designer: *Lumina Datamatics Ltd.*
Cover Photo: *George Spade/123RF*
Manager, Rights and Permissions: *Rachel Youdelman*
Photo Permission Coordinator: *Rachel Youdelman*
Image Permission Coordinator: *Paul Sarkis*
Full-Service Project Management: *Jouve India*
Head of Learning Asset Acquisition, Global Edition: *Laura Dent*
Senior Manufacturing Controller, Global Edition: *Trudy Kimber*
Senior Acquisitions Editor, Global Edition: *Priyanka Ahuja*
Project Editor, Global Edition: *Aaditya Bugga*

Pearson Education Limited
Edinburgh Gate
Harlow
Essex CM20 2JE
England

and Associated Companies throughout the world

Visit us on the World Wide Web at:
www.pearsonglobaleditions.com

Authorized adaptation from the United States edition, entitled Finite Element Analysis: Theory and Application with ANSYS, 4th edition, ISBN 978-0-13-384080-3, by Saeed Moaveni, published by Pearson Education © 2015.

ISBN 10: 0-273-77430-1
ISBN 13: 978-0-273-77430-3

British Library Cataloguing-in-Publication Data
A catalogue record for this book is available from the British Library

10 9 8 7 6 5 4 3 2 1
19 18 17 16 15

Typeset in 10 Times Ten LT Std by Jouve India.

Printed in Great Britain by CPI Group (UK) Ltd, Croydon, CRO 4YY

To memories of my mother and father

Contents

Preface

CHANGES IN THE FOURTH EDITION

The fourth edition, consisting of 15 chapters, includes a number of new additions and changes that were incorporated in response to ANSYS revisions and suggestions and requests made by professors, students, and professionals using the third edition of the book. The major changes include:

- Explanation of the changes that were made in the ANSYS's newest release (Chapters 3 and 8)
- Explanation of new element type capabilities (Chapters 3, 4, 6, 8 through 13, and 15)
- A new comprehensive example problem that demonstrates the use of BEAM188 element in modeling beam and frame problems (Chapter 4)
- Modification of twenty example problems to incorporate new ANSYS element types (Chapters 3, 4, 6, 8 through 13, and 15)
- Eight new comprehensive example problems that show in great detail how to use Excel to solve different types of finite element problems (Chapters 2 through 6 and 9 through 12)
- More detail on theory and expanded derivations
- Explanation of new MATLAB revisions in Appendix F

ORGANIZATION

There are many good textbooks already in existence that cover the theory of finite element methods for advanced students. However, none of these books incorporate ANSYS as an integral part of their materials to introduce finite element modeling to undergraduate students and newcomers. In recent years, the use of finite element analysis (FEA) as a design tool has grown rapidly. Easy-to-use, comprehensive packages such as ANSYS, a general-purpose finite element computer program, have become common tools in the hands of design engineers. Unfortunately, many engineers who lack the proper training or understanding of the underlying concepts have been using these tools. This introductory book is written to assist engineering students and practicing engineers new to the field of finite element modeling to gain a clear understanding of the basic concepts. The text offers insight into the theoretical aspects of FEA and also covers some practical aspects of modeling. Great care has been exercised to avoid overwhelming students with theory, yet enough theoretical background is offered to allow individuals to use ANSYS intelligently and effectively. ANSYS is an integral part of this text. In each chapter, the relevant basic theory is discussed first and demonstrated using simple problems with hand calculations. These problems are followed by examples that are solved using ANSYS. Exercises in the text are also presented in this manner. Some exercises require manual calculations, while others, more complex in nature, require the use of ANSYS. The simpler hand-calculation problems will enhance students' understanding of the concepts by encouraging them to go through the necessary steps in a FEA. Design problems are also included at the end of Chapters 3, 4, 6, and 9 through 14.

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

Another unique feature of this book is that the last two chapters are devoted to the introduction of design, material selection, optimization, and parametric programming with ANSYS.

The book is organized into 15 chapters. Chapter 1 reviews basic ideas in finite element analysis. Common formulations, such as direct, potential energy, and weighted residual methods, are discussed. Chapter 2 provides a comprehensive review of matrix algebra. Chapter 3 deals with the analysis of trusses, because trusses offer economical solutions to many engineering structural problems. An overview of the ANSYS program is given in Chapter 3 so that students can begin to use ANSYS right away. Finite element formulation of members under axial loading, beams, and frames are introduced in Chapter 4. Chapter 5 lays the foundation for analysis of one-dimensional problems by introducing one-dimensional linear, quadratic, and cubic elements. Global, local, and natural coordinate systems are also discussed in detail in Chapter 5. An introduction to isoparametric formulation and numerical integration by Gauss–Legendre formulae is also presented in Chapter 5. Chapter 6 considers Galerkin formulation of one-dimensional heat transfer and fluid problems. Two-dimensional linear and higher order elements are introduced in Chapter 7. Gauss–Legendre formulae for two-dimensional integrals are also presented in Chapter 7. In Chapter 8 the essential capabilities and the organization of the ANSYS program are covered. The basic steps in creating and analyzing a model with ANSYS is discussed in detail. Chapter 9 includes the analysis of two-dimensional heat transfer problems with a section devoted to unsteady situations. Chapter 10 provides an analysis of torsion of noncircular shafts and plane stress problems. Dynamic problems are explored in Chapter 11. Review of dynamics and vibrations of mechanical and structural systems are also given in this chapter. In Chapter 12, two-dimensional, ideal fluid-mechanics problems are analyzed. Direct formulation of the piping network problems and underground seepage flow are also discussed. Chapter 13 provides a discussion on three-dimensional elements and formulations. This chapter also presents basic ideas regarding top-down and bottom-up solid modeling methods. The last two chapters of the book are devoted to design and optimization ideas. Design process and material selection are explained in Chapter 14. Design optimization ideas and parametric programming are discussed in Chapter 15. Examples of ANSYS batch files are also given in Chapter 15. Each chapter begins by stating the objectives and concludes by summarizing what the reader should have gained from studying that chapter.

The examples that are solved using ANSYS show in great detail how to use ANSYS to model and analyze a variety of engineering problems. Chapter 8 is also written such that it can be taught right away if the instructor sees the need to start with ANSYS.

A brief review of appropriate fundamental principles in solid mechanics, heat transfer, dynamics, and fluid mechanics is also provided throughout the book. Additionally, when appropriate, students are warned about becoming too quick to generate finite element models for problems for which there exist simple analytical solutions. Mechanical and thermophysical properties of some common materials used in engineering are given in Appendices A and B. Appendices C and D give properties of common area shapes and properties of structural steel shapes, respectively. A comprehensive introduction to MATLAB is given in Appendix F.

Finally, a Web site at *http://www.pearsonglobaleditions.com/moaveni* will be maintained for the following purposes: (1) to share any changes in the upcoming versions of ANSYS; (2) to share additional information on upcoming text revisions; (3) to provide additional homework problems and design problems; and (4) although I have done my best to eliminate errors and mistakes, as is with most books, some errors may still exist. I will post the corrections that are brought to my attention at the site. The Web site will be accessible to all instructors and students.

Thank you for considering this book and I hope you enjoy the fourth edition.

Saeed Moaveni

Acknowledgments

I would like to express my sincere gratitude to ANSYS, Inc. for providing the photographs that appear on page 27 of this book. Descriptions for these photographs are given in Chapter 1 with the images. I would also like to thank ANSYS, Inc. for giving me permission to adapt material from various ANSYS documents, related to capabilities and the organization of ANSYS. The essential capabilities and organizations of ANSYS are covered in Chapters 3, 8, 13, and 15.

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

I am also thankful to all reviewers who offered general and specific comments.

GLOBAL EDITION

Pearson would like to thank and acknowledge the following people for their work on the Global Edition:

CONTRIBUTOR

Jagadeesha T, National Institute of Technology—Calicut

REVIEWERS

Swarup Bag, Indian Institute of Technology—Guwahati
Avinash Parashar, Indian Institute of Technology—Roorkee
Mokhtar B Awang, Universiti Teknologi Petronas—Malaysia
Yash Parikh, Symbiosis Institute of Technology—Pune

FINITE ELEMENT ANALYSIS

CHAPTER 1

Introduction

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

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

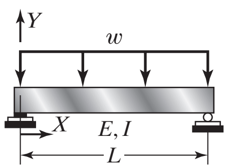
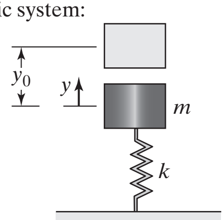
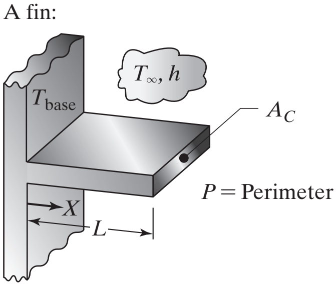
The objective of this chapter is to introduce you to basic concepts in finite element formulation, including direct formulation, the minimum potential energy theorem, and the weighted residual methods. The main topics of Chapter 1 include the following:

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

1.1 ENGINEERING PROBLEMS

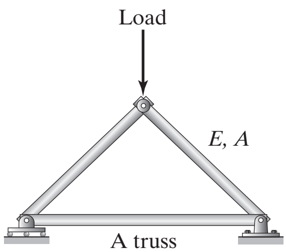
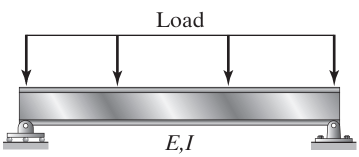
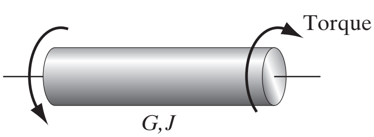
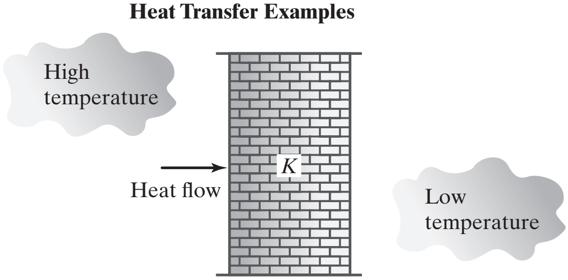
In general, engineering problems are mathematical models of physical situations. Mathematical models of many engineering problems are differential equations with a set of corresponding boundary and/or initial conditions. The differential equations are derived by applying the fundamental laws and principles of nature to a system or a control volume. These governing equations represent balance of mass, force, or energy. When possible, the exact solution of these equations renders detailed behavior of a system under a given set of conditions, as shown by some examples in Table 1.1. The analytical solutions are composed of two parts: (1) a homogenous part and (2) a particular part. In any given engineering problem, there are two sets of design parameters that influence the way in which a system behaves. First, there are those parameters that

TABLE 1.1 Examples of governing differential equations, boundary conditions, initial conditions, and exact solutions for some engineering problems

Problem Type	Governing Equation, Boundary Conditions, or Initial Conditions	Solution
A beam: 	$EI\frac{d^2Y}{dX^2} = \frac{wX(L - X)}{2}$ Boundary conditions: at $X = 0$, $Y = 0$ and at $X = L$, $Y = 0$	Deflection of the beam Y as the function of distance X: $Y = \frac{w}{24EI}(-X^4 + 2LX^3 - L^3X)$
An elastic system: 	$\frac{d^2y}{dt^2} + \omega_n^2 y = 0$ where $\omega_n^2 = \frac{k}{m}$ Initial conditions: at time $t = 0$, $y = y_0$ and at time $t = 0$, $\frac{dy}{dt} = 0$	The position of the mass y as the function of time: $y(t) = y_0 \cos \omega_n t$
A fin: 	$\frac{d^2T}{dX^2} - \frac{hp}{kA_c}(T - T_\infty) = 0$ Boundary conditions: at $X = 0$, $T = T_{\text{base}}$ as $L \to \infty$, $T = T_\infty$	Temperature distribution along the fin as the function of X: $T = T_\infty + (T_{\text{base}} - T_\infty)e^{-\sqrt{\frac{hp}{kA_c}}X}$

provide information regarding the *natural behavior* of a given system. These parameters include material and geometric properties such as modulus of elasticity, thermal conductivity, viscosity, and area, and second moment of area. Table 1.2 summarizes the physical properties that define the natural characteristics of various problems.

TABLE 1.2 Physical properties characterizing various engineering systems

Problem Type	Examples of Parameters That Characterize a System
Solid Mechanics Examples	
 A truss	Modulus of elasticity, E; member length, L; cross-sectional area, A
 An elastic plate	Modulus of elasticity, E; length, L; cross-sectional area, A
 A beam	Modulus of elasticity, E; member length, L; second moment of area, I
 A shaft	Modulus of rigidity, G; member length, L; polar moment of inertia of the area, J
Heat Transfer Examples	
 A wall	Thermal conductivity, K; thickness, L; area, A

continued

TABLE 1.2 *Continued*

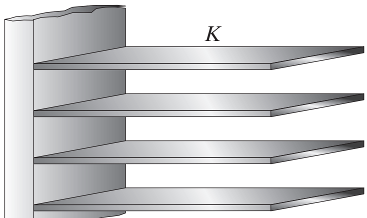
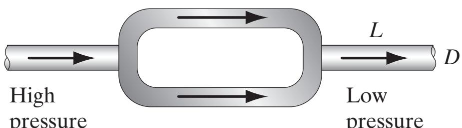
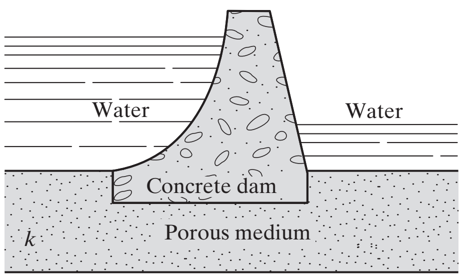
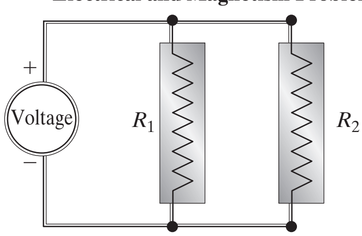
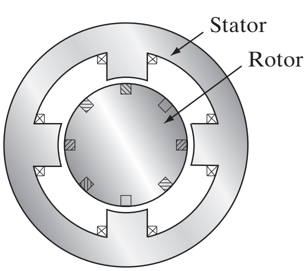
Problem Type	Examples of Parameters That Characterize a System
 Fins	Thermal conductivity, K; perimeter, P; cross-sectional area, A
Fluid Flow Examples  Pipe networks	Fluid viscosity, μ; pipe roughness, e; pipe diameter, D; pipe length, L
 A concrete dam	Soil permeability, k
Electrical and Magnetism Problems  Electrical network	Resistance, R
 Magnetic field of an electric motor	Permeability, μ

TABLE 1.3 Parameters causing disturbances in various engineering systems

Problem Type	Examples of Parameters that Produce Disturbances in a System
Solid Mechanics	External forces and moments; support excitation
Heat Transfer	Temperature difference; heat input
Fluid Flow and Pipe Networks	Pressure difference; rate of flow
Electrical Network	Voltage difference

On the other hand, there are parameters that produce *disturbances* in a system. These types of parameters are summarized in Table 1.3. Examples of these parameters include external forces, moments, temperature difference across a medium, and pressure difference in fluid flow.

The system characteristics as shown in Table 1.2 dictate the natural behavior of a system, and they always appear in the *homogenous part of the solution* of a governing differential equation. In contrast, the parameters that cause the disturbances appear in the *particular solution*. It is important to understand the role of these parameters in finite element modeling in terms of their respective appearances in stiffness or conductance matrices and load or forcing matrices. The system characteristics will always show up in the stiffness matrix, conductance matrix, or resistance matrix, whereas the disturbance parameters will always appear in the load matrix. We will explain the concepts of stiffness, conductance, and load matrices in Section 1.5.

1.2 NUMERICAL METHODS

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

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

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

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

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

ANSYS is a comprehensive general-purpose finite element computer program that contains more than 100,000 lines of code. ANSYS is capable of performing static, dynamic, heat transfer, fluid flow, and electromagnetism analyses. ANSYS has been a leading FEA program for over 40 years. The current version of ANSYS has a completely new look, with multiple windows incorporating a graphical user interface (GUI), pull-down menus, dialog boxes, and a tool bar. Today, you will find ANSYS in use in many engineering fields, including aerospace, automotive, electronics, and nuclear. In order to use ANSYS or any other "canned" FEA computer program intelligently, it is imperative that one first fully understands the underlying basic concepts and limitations of the finite element methods.

ANSYS is a very powerful and impressive engineering tool that may be used to solve a variety of problems (see Table 1.4). However, a user without a basic understanding of the finite element methods will find himself or herself in the same predicament as a computer technician with access to many impressive instruments and tools, but who cannot fix a computer because he or she does not understand the inner workings of a computer!

1.4 BASIC STEPS IN THE FINITE ELEMENT METHOD

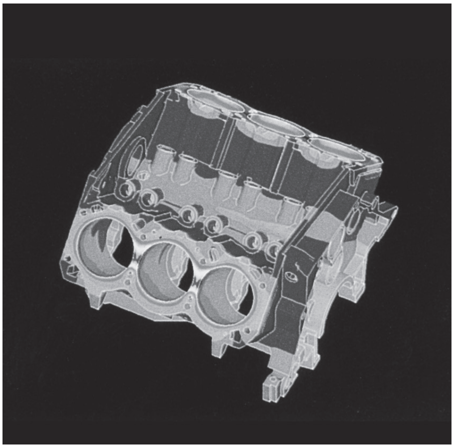
The basic steps involved in any finite element analysis consist of the following:

Preprocessing Phase

1. Create and discretize the solution domain into finite elements; that is, subdivide the problem into nodes and elements.

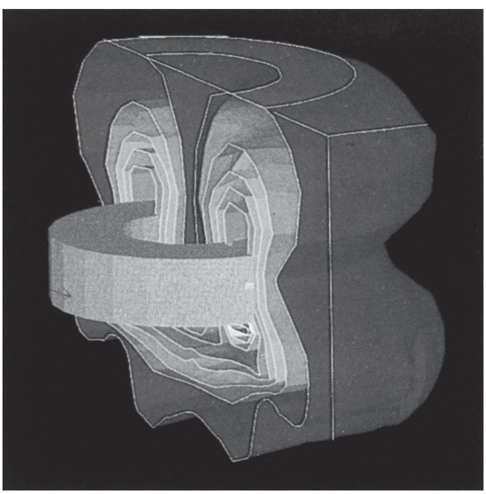
*See Cook et al. (1989) for more detail.

TABLE 1.4 Examples of the capabilities of ANSYS*

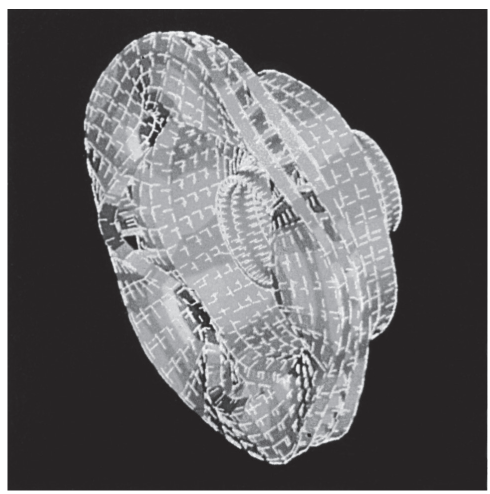


A V6 engine used in front-wheel-drive automobiles analyses were conducted by Analysis & Design Appl. Co. Ltd. (ADAPCO) on behalf of a major U.S. automobile manufacturer to improve product performance. Contours of thermal stress in the engine block are shown in the figure above.

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



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



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

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

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

Solution Phase

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

Postprocessing Phase

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

In general, there are several approaches to formulating finite element problems: (1) *direct formulation,* (2) *the minimum total potential energy formulation,* and (3) *weighted residual formulations*. Again, it is important to note that the basic steps involved in any finite element analysis, regardless of how we generate the finite element model, will be the same as those listed above.

1.5 DIRECT FORMULATION

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

EXAMPLE 1.1

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

Preprocessing Phase

1. *Discretize the solution domain into finite elements.*
   We begin by subdividing the problem into nodes and elements. In order to highlight the basic steps in a finite element analysis, we will keep this problem simple and thus represent it by a model that has five nodes and four elements, as shown in Figure 1.2. However, note that we can increase the accuracy of our results by generating a model with additional nodes and elements. This task is left as an exercise for you to complete. (See Problem 1 at the end of this chapter.) The given bar

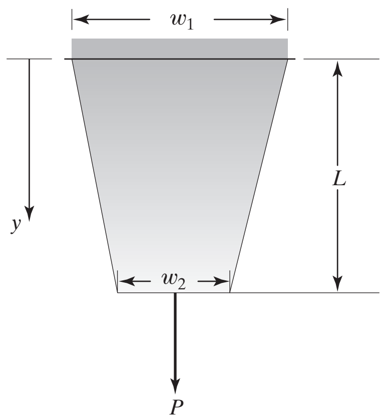


FIGURE 1.1 A bar under axial loading.

is modeled using four individual segments (elements), with each segment having a uniform cross section. The cross-sectional area of each element is represented by an average area of the cross sections at the nodes that make up (define) the element. This model is shown in Figure 1.2.

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

   The average stress σ in the member is given by

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

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

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

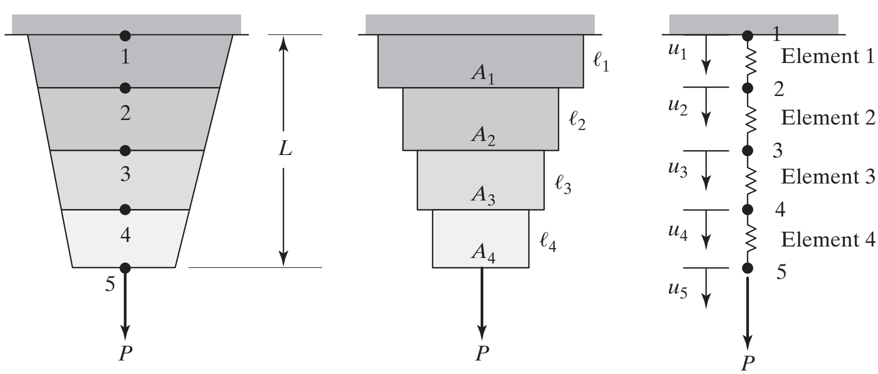


FIGURE 1.2 Subdividing the bar into elements and nodes.

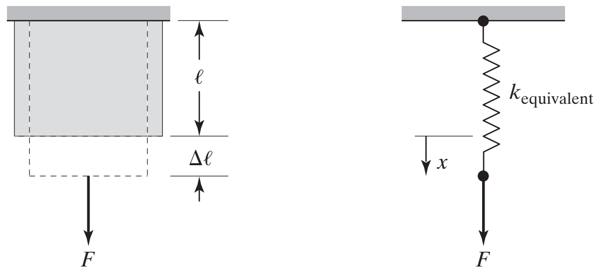


FIGURE 1.3 A solid member of uniform cross section subjected to a force F.

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

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

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

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

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

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

Turning our attention to Example 1.1, we note once again that the bar's cross section varies in the y-direction. As a first approximation, we model the bar as a series of centrally loaded members with different cross sections, as shown in Figure 1.2. Thus, the bar is represented by a model consisting of four elastic springs (elements) in series, and the elastic behavior of an element with nodes i and $i + 1$ is modeled by an equivalent linear spring according to the equation

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

where u_{i+1} and u_i are the deflections at nodes $i + 1$ and i, and the equivalent element stiffness is given by

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

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