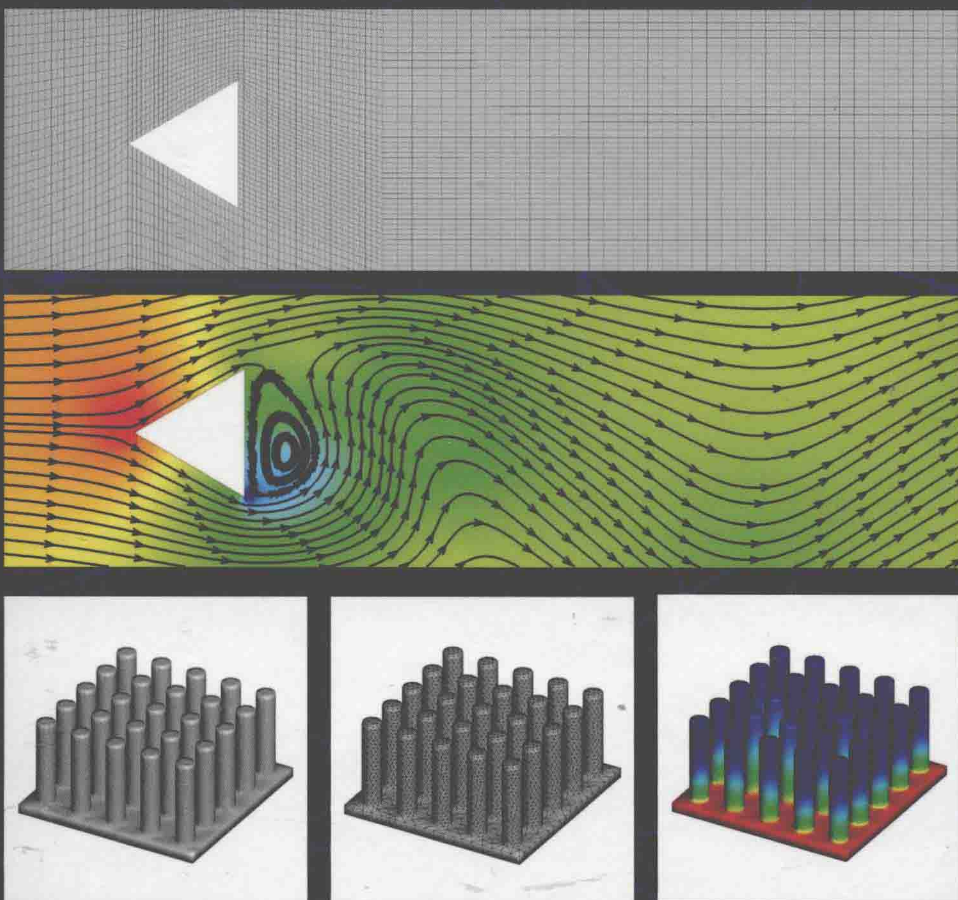
Finite Element Simulations Using ANSYS

Second Edition



Esam M. Alawadhi

CRC Press
Taylor & Francis Group

Finite Element Simulations Using ANSYS

Second Edition

Esam M. Alawadhi
Kuwait University
Safat, Kuwait

CRC Press
Taylor & Francis Group
Boca Raton London New York

CRC Press is an imprint of the
Taylor & Francis Group, an **informa** business

CRC Press
Taylor & Francis Group
6000 Broken Sound Parkway NW, Suite 300
Boca Raton, FL 33487-2742

Printed on acid-free paper
Version Date: 20150728

International Standard Book Number-13: 978-1-4822-6197-4 (Hardback)

Visit the Taylor & Francis Web site at
http://www.taylorandfrancis.com

and the CRC Press Web site at
http://www.crcpress.com

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

Finite Element Simulations Using ANSYS

Second Edition

Preface

Due to the complexity of modern-day problems in mechanical engineering, engineers, in practice, seldom rely only on theory or experiments. The use of engineering software is becoming prevalent among academics as well as practicing engineers. Commercial finite element software such as ANSYS (ANSYS, Canonsburg, Pennsylvania) and Abaqus (Dassault Systèmes, Vélizy Villacoublay, France) for analysis and design is commonly found in use in universities, industry, and research centers. This software has become more reliable, reputable, easy to use, and trustworthy, and it allows much time to be saved.

This book focuses on the use of ANSYS in solving practical mechanical engineering problems. ANSYS is extensively used in the design cycle by industry leaders in the United States and around the world. Additionally, ANSYS is available in computer labs for students in most universities around the world. Courses such as computer-aided design, modeling, and simulation, and major design courses all utilize ANSYS as a tool for analyzing various mechanical components. Graduate students also use ANSYS in their finite element courses as a complement for the theoretical study of the finite element method.

This book provides mechanical engineering students and engineers with the fundamental knowledge of numerical simulation using ANSYS. The book serves most of the disciplines of mechanical engineering: structure, solid mechanics, vibration, heat transfer, and fluid dynamics, with adequate background material to explain the physics behind the computations. Each physical phenomenon is treated independently in a way that enables readers to pick out a single subject or a related chapter and study it. Instructors can cover appropriate chapters depending on the objectives of the course. The required basic knowledge of the finite element method relevant to each physical phenomenon is illustrated at the beginning of the respective chapter. The general theory of the finite element, however, is presented briefly and concisely because the theory is well documented by other finite element books.

For example, in the heat transfer chapter, the theory is first explained, the governing equations are derived, the modeling techniques are presented, and finally practical problems are solved using ANSYS in a step-by-step technique. Each chapter independently discusses a single physical phenomenon, while the last chapter is devoted to multiphysics analyses and problems. The finite element solution is greatly affected by the quality of the mesh, and therefore, a separate chapter on meshing is included as a guide that emphasizes the basics of the meshing techniques. Practical end-of-chapter problems are provided in each chapter to challenge the reader's understanding.

Undergraduate and graduate engineers will use this book as a part of their courses, either when studying the basics of applied finite elements, or in mastering practical tools of engineering modeling. Engineers in industry can use this book as a guide for better design and analysis of their products. In all mechanical engineering curricula, junior- and senior-level courses use some type of engineering modeling software, and, oftentimes, this software is ANSYS. Senior students also use ANSYS in their senior design projects. Graduate-level finite element courses frequently use ANSYS to complement the theoretical analysis of finite elements. The course that uses this book should be taken after the introduction to design courses and the basic thermal–fluid courses. Courses such as that in senior design can be taken after this course.

In this second edition of the book, new sections are added, and ANSYS examples are modified to be in compliance with the new version of ANSYS. Most ANSYS examples in the first edition are replaced by more general, comprehensive, and easy-to-follow examples. In the finite element theoretical part, more details are added, especially for the heat transfer chapter. Additionally, open-ended problems are added at the end of each chapter, which can serve as class projects.

Acknowledgments

I profited greatly from discussion with faculty members and engineers at Kuwait University. I want to mention particularly Professor Ahmed Yigit.

About the author

Esam M. Alawadhi is a professor of mechanical engineering at Kuwait University, Kuwait City, Kuwait. He earned his doctor of philosophy in mechanical engineering from Carnegie Mellon University, Pittsburgh, Pennsylvania in May 2001. His research focuses on renewable energy, thermal management of electronics devices, energy conservation for buildings, fluid flow stability, and phase-change heat transfer.

Contents

CHAPTER ONE

Introduction to finite element method and ANSYS

1.1 The finite element method and structural analysis

The finite element method was developed in the 1950s and has been continuously enhanced since then. Rapid advances in computing power and the drastic drop in cost make the finite element method affordable. Now, it is a commonly used method for solving a wide range of problems, and the potential of the finite element method is enormous. The finite element method is typically found in the aerospace, automotive, electrical, hydraulic, biomedical, nuclear, and structural engineering fields, among many others. The first step in the finite element solution procedure is to divide the domain into elements, and this process is called the *domain discretization*. The variable distribution across each element can be defined by linear, quadratic, or trigonometric function. The elements distribution in the domain is called the *finite element mesh*. The elements are connected at points called *nodes*. For example, consider a section of a bridge, as shown in Figure 1.1a. The bridge is divided into linear elements and connected by nodes, as shown in Figure 1.1b.

After the region is discretized, the governing equations for the element must be established for the required physics. Material properties, such as modulus of elasticity for structural analysis, should be available. The equations are assembled to obtain the global equation for the mesh, which describes the behavior of the body as a whole. Generally, the global governing equation has the following form:

$$[K]\{U\} = \{F\} \tag{1.1}$$

where [K] is called the stiffness matrix; {U} is the nodal degree of freedom, such as the displacements for structural analysis; and {F} is the nodal external force, such as forces for structural analysis. The [K] matrix is a singular matrix, and therefore it cannot be inverted.

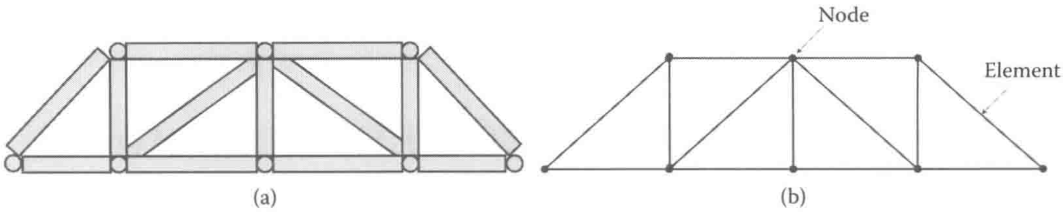


FIGURE 1.1 (a) A section of a bridge and (b) finite element mesh.

Consider a one-dimensional bar with initial length L that is subjected to a tensile force at its ends, as shown in Figure 1.2a. The cross-sectional area of the bar is A. The element can be modeled using a single element with two nodes, i and j, as shown in Figure 1.2b.

Assume that the displacement function of the bar u(x) varies linearly along the length of the bar. The expression of the displacement is represented as

$$u(x) = a + bx \tag{1.2}$$

The displacement at nodes i and j are u_i and u_j, respectively. Then

$$u_i(x) = a + bx_i \tag{1.3}$$

$$u_j(x) = a + bx_j \tag{1.4}$$

where x_i is the x-coordinate for node i, and x_j is the x-coordinate for node j. Solving for constants a and b, it is found that

$$a = (u_i x_j - u_j x_i)/L \tag{1.5}$$

$$b = (u_j - u_i) / L \tag{1.6}$$

where L is the initial length of the element and is equal to $(x_i - x_j)$. Substituting constants a and b into the displacement equation (Equation 1.2) and rearranging, the displacement function becomes

$$u(x) = \frac{x_j - x}{L} u_i + \frac{x - x_i}{L} u_j \tag{1.7}$$

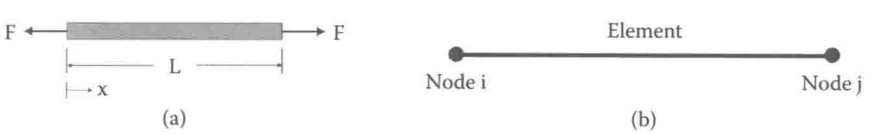


FIGURE 1.2 A one-dimensional bar element. (a) A single one-dimensional bar element and (b) an element with two nodes.