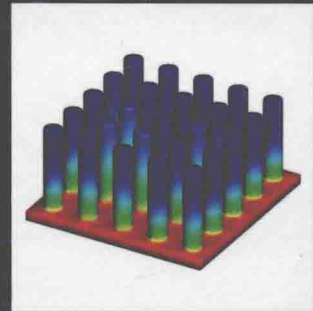
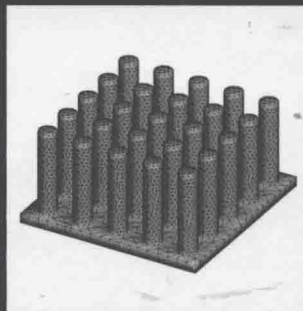
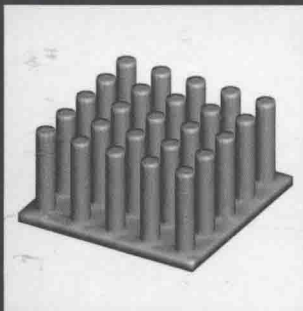
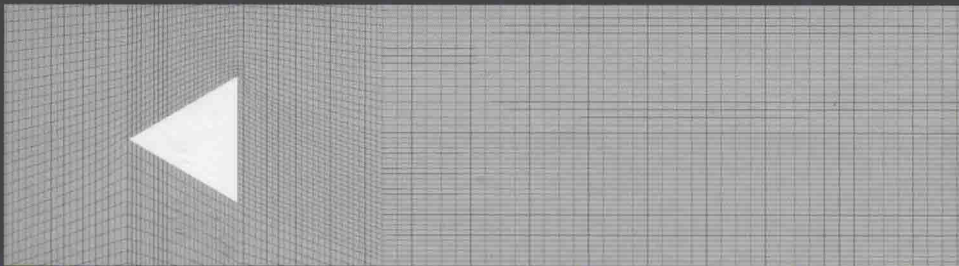


Finite Element Simulations Using ANSYS

Second Edition



Esam M. Alawadhi



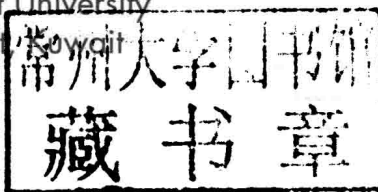
CRC Press
Taylor & Francis Group

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

© 2016 by Taylor & Francis Group, LLC
CRC Press is an imprint of Taylor & Francis Group, an Informa business

No claim to original U.S. Government works

Printed on acid-free paper
Version Date: 20150728

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

This book contains information obtained from authentic and highly regarded sources. Reasonable efforts have been made to publish reliable data and information, but the author and publisher cannot assume responsibility for the validity of all materials or the consequences of their use. The authors and publishers have attempted to trace the copyright holders of all material reproduced in this publication and apologize to copyright holders if permission to publish in this form has not been obtained. If any copyright material has not been acknowledged please write and let us know so we may rectify in any future reprint.

Except as permitted under U.S. Copyright Law, no part of this book may be reprinted, reproduced, transmitted, or utilized in any form by any electronic, mechanical, or other means, now known or hereafter invented, including photocopying, microfilming, and recording, or in any information storage or retrieval system, without written permission from the publishers.

For permission to photocopy or use material electronically from this work, please access www.copyright.com (<http://www.copyright.com/>) or contact the Copyright Clearance Center, Inc. (CCC), 222 Rosewood Drive, Danvers, MA 01923, 978-750-8400. CCC is a not-for-profit organization that provides licenses and registration for a variety of users. For organizations that have been granted a photocopy license by the CCC, a separate system of payment has been arranged.

Trademark Notice: Product or corporate names may be trademarks or registered trademarks, and are used only for identification and explanation without intent to infringe.

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

<i>Preface</i>	ix
<i>Acknowledgments</i>	xi
<i>About the author</i>	xiii

1	Introduction to finite element method and ANSYS	1
1.1	The finite element method and structural analysis	1
1.2	Stress analysis of pin-jointed bar	5
1.3	The finite element method and thermal analysis	6
1.4	Heat transfer through a composite wall	8
1.5	Introduction to ANSYS software	10
2	Bar and beam structures	19
2.1	Finite element method for a horizontal bar element	19
2.2	Finite element method for an arbitrary-oriented bar element	23
2.3	Analyzing a plane bar structure	26
2.4	Analyzing a plane bar structure using ANSYS	29
2.5	Finite element method for a horizontal beam element	42
2.6	Analyzing a horizontal beam structure	49
2.7	Analyzing a horizontal beam structure using ANSYS	51
2.8	Development of an arbitrary-oriented beam element	67
2.9	Distributed load on a beam element	70
2.10	Analyzing beam structure under a transient loading using ANSYS	72

3 Solid mechanics	95
3.1 Stress–strain relations	95
3.2 Development of triangular plane stress and plane strain element	100
3.3 Analyzing rectangular plate subjected to forces	105
3.4 Development of rectangular plane stress element	109
3.5 Analyzing a plate with a hole subjected to tensile pressure using ANSYS	112
3.6 Axisymmetric elements	126
3.7 Displacement analysis of a vessel under transient loading with ANSYS	130
3.8 Contact element analysis	145
3.9 Two horizontal cylinders in contact analysis using ANSYS	147
4 Vibration	171
4.1 Vibration analysis	171
4.2 Development of stiffness equations for axial vibration of a bar element	172
4.3 Natural frequencies of axial vibration of a bar element	173
4.4 Development of stiffness equations for flexural vibration of a beam element	175
4.5 Natural frequencies of the flexural vibration of a beam element	177
4.6 Natural frequencies of the flexural vibration of beam element using ANSYS	179
4.7 Development of stiffness equations for vibration of an oriented beam element	190
4.8 Harmonic vibration of a plate with holes using ANSYS	192
4.9 Three-dimensional vibration of shaft with disks using ANSYS	207
5 Heat transfer	225
5.1 Introduction to heat conduction	225
5.2 Finite element formulation for conductive heat transfer	228
5.3 Finite element method for one-dimensional heat conduction	231
5.4 Heat transfer through a composite wall	233
5.5 Finite element method for two-dimensional heat conduction	235
5.6 Heat conduction in a solid plane	238
5.7 Thermal analysis of fin and chip using ANSYS	243
5.8 Finite element method for transient heat transfer	264
5.9 Unsteady thermal analyses of a masonry brick using ANSYS	265

6 Fluid mechanics	293
6.1 Governing equations for fluid mechanics	293
6.2 Finite element method for fluid mechanics	296
6.3 Entrance length in developing flow in a channel using ANSYS	298
6.4 Studying flow around a half cylinder in a channel using ANSYS	318
7 Multiphysics	343
7.1 Introduction	343
7.2 Thermal and structural analysis of a thermocouple using ANSYS	345
7.3 Chips cooling in a channel using ANSYS	361
8 Meshing guide	393
8.1 Mesh refinement	393
8.2 Element distortion	395
8.3 Mapped mesh	396
8.4 Mapped mesh with ANSYS	398
Bibliography	411
Index	413

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:

$$[K]\{U\} = \{F\} \quad (1.1)$$

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.

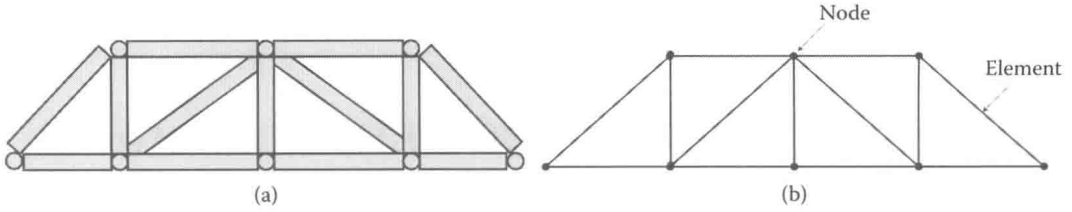


FIGURE 1.1 (a) A section of a bridge and (b) finite element mesh.

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

$$u(x) = a + bx \quad (1.2)$$

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

$$u_i(x) = a + bx_i \quad (1.3)$$

$$u_j(x) = a + bx_j \quad (1.4)$$

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

$$a = (u_i x_j - u_j x_i) / L \quad (1.5)$$

$$b = (u_j - u_i) / L \quad (1.6)$$

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

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



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