The Finite Element Method and Applications in Engineering Using ANSYS®

Erdogan Madenci • Ibrahim Guven

The Finite Element Method and Applications in Engineering Using ANSYS®

Second Edition

Springer is a brand of Springer US

Springer

Erdogan Madenci
Department of Aerospace and Mechanical Engineering
The University of Arizona
Tucson, Arizona
USA

Ibrahim Guven
Department of Mechanical and Nuclear Engineering
Virginia Commonwealth University
Richmond, Virginia
USA

Reproduction of all Copyrighted Material of ANSYS software and GUI was with the permission of ANSYS, Inc. ANSYS, Inc. product names are trademarks or registered trademarks of ANSYS, Inc. or its subsidiaries in the United States or other countries.

Electronic supplementary material can be found at http://link.springer.com/book/10.1007/978-1-4899-7550-8.

ISBN 978-1-4899-7549-2 ISBN 978-1-4899-7550-8 (eBook)
DOI 10.1007/978-1-4899-7550-8

Library of Congress Control Number: 2014955722

Springer New York Heidelberg Dordrecht London
© Springer International Publishing 2015
This work is subject to copyright. All rights are reserved by the Publisher, whether the whole or part of the material is concerned, specifically the rights of translation, reprinting, reuse of illustrations, recitation, broadcasting, reproduction on microfilms or in any other physical way, and transmission or information storage and retrieval, electronic adaptation, computer software, or by similar or dissimilar methodology now known or hereafter developed.
The use of general descriptive names, registered names, trademarks, service marks, etc. in this publication does not imply, even in the absence of a specific statement, that such names are exempt from the relevant protective laws and regulations and therefore free for general use.
The publisher, the authors and the editors are safe to assume that the advice and information in this book are believed to be true and accurate at the date of publication. Neither the publisher nor the authors or the editors give a warranty, express or implied, with respect to the material contained herein or for any errors or omissions that may have been made.

Printed on acid-free paper

Springer is part of Springer Science+Business Media (www.springer.com)

Preface

The finite element method (FEM) has become a staple for predicting and simulating the physical behavior of complex engineering systems. The commercial finite element analysis (FEA) programs have gained common acceptance among engineers in industry and researchers at universities and government laboratories. Therefore, academic engineering departments include graduate or undergraduate senior-level courses that cover not only the theory of FEM but also its applications using the commercially available FEA programs.

The goal of this book is to provide students with a theoretical and practical knowledge of the finite element method and the skills required to analyze engineering problems with ANSYS®, a commercially available FEA program. This book, designed for seniors and first-year graduate students, as well as practicing engineers, is introductory and self-contained in order to minimize the need for additional reference material.

In addition to the fundamental topics in finite element methods, it presents advanced topics concerning modeling and analysis with ANSYS®. These topics are introduced through extensive examples in a step-by-step fashion from various engineering disciplines. The book focuses on the use of ANSYS® through both the Graphics User Interface (GUI) and the ANSYS® Parametric Design Language (APDL). Furthermore, it includes a CD-ROM with the "*input*" files for the example problems so that the students can regenerate them on their own computers. Because of printing costs, the printed figures and screen shots are all in gray scale. However, color versions are provided on the accompanying CD-ROM.

Chapter 1 provides an introduction to the concept of FEM. In Chap. 2, the analysis capabilities and fundamentals of ANSYS®, as well as practical modeling considerations, are presented. The fundamentals of discretization and approximation functions are presented in Chap. 3. The modeling techniques and details of mesh generation in ANSYS® are presented in Chap. 4. Steps for obtaining solutions and reviews of results are presented in Chap. 5. In Chap. 6, the derivation of finite element equations based on the method of weighted residuals and principle of minimum potential energy is explained and demonstrated through example problems. The use of commands and APDL and the development of macro files are presented in Chap. 7. In Chap. 8, example problems on linear structural analysis are worked

out in detail in a step-by-step fashion. The example problems related to heat transfer and moisture diffusion are demonstrated in Chap. 9. Nonlinear structural problems are presented in Chap. 10. Advanced topics concerning submodeling, substructuring, interaction with external files, and modification of ANSYS®-GUI are presented in Chap. 11.

There are more than 40 example problems considered in this book; solutions to most of these problems using ANSYS® are demonstrated using GUI in a step-by-step fashion. The remaining problems are demonstrated using the APDL. However, the steps taken in either GUI- or APDL-based solutions may not be the optimum/shortest possible way. Considering the steps involved in obtaining solutions to engineering problems (e.g., model generation, meshing, solution options, etc.), there exist many different routes to achieve the same solution. Therefore, the authors strongly encourage the students/engineers to experiment with modifications to the analysis steps presented in this book.

We are greatly indebted to Connie Spencer for her invaluable efforts in typing, editing, and assisting with each detail associated with the completion of this book. Also, we appreciate the contributions made by Dr. Atila Barut, Dr. Erkan Oterkus, Dr. Abigail Agwai, Dr. Manabendra Das, and Dr. Bahattin Kilic in the solution of the example problems. Last, but not least, we thank Mr. Mehmet Dorduncu for his careful review of the modeling steps and example problems, and for capturing the ANSYS screen shots in this version of the book. The permission provided by ANSYS, Inc. to print the screen shots is also appreciated.

Contents

1 Introduction 1
- 1.1 Concept 1
- 1.2 Nodes 3
- 1.3 Elements 3
- 1.4 Direct Approach 4
  - 1.4.1 Linear Spring 5
  - 1.4.2 Heat Flow 6
  - 1.4.3 Assembly of the Global System of Equations 7
  - 1.4.4 Solution of the Global System of Equations 11
  - 1.4.5 Boundary Conditions 12

2 Fundamentals of ANSYS 15
- 2.1 Useful Definitions 15
- 2.2 Before an ANSYS Session 16
  - 2.2.1 Analysis Discipline 16
  - 2.2.2 Time Dependence 17
  - 2.2.3 Nonlinearity 18
  - 2.2.4 Practical Modeling Considerations 19
- 2.3 Organization of ANSYS Software 25
- 2.4 ANSYS Analysis Approach 25
  - 2.4.1 ANSYS Preprocessor 26
  - 2.4.2 ANSYS Solution Processor 26
  - 2.4.3 ANSYS General Postprocessor 26
  - 2.4.4 ANSYS Time History Postprocessor 26
- 2.5 ANSYS File Structure 26
  - 2.5.1 Database File 27
  - 2.5.2 Log File 27
- 2.6 Error File 27
  - 2.6.1 Results Files 28
- 2.7 Description of ANSYS Menus and Windows 28
  - 2.7.1 Utility Menu 28
  - 2.7.2 Main Menu 30

2.7.3 Toolbar 30
2.7.4 Input Field 30
2.7.5 Graphics Window 30
2.7.6 Output Window 30
2.8 Using the ANSYS Help System 31
2.8.1 Help Contents 31
2.8.2 Help Index 32
2.8.3 Search in Help 32
2.8.4 Verification Manual 33

3 Fundamentals of Discretization 35
3.1 Local and Global Numbering 35
3.2 Approximation Functions 35
3.3 Coordinate Systems 40
3.3.1 Generalized Coordinates 40
3.3.2 Global Coordinates 40
3.3.3 Local Coordinates 40
3.3.4 Natural Coordinates 41
3.4 Shape Functions 47
3.4.1 Linear Line Element with Two Nodes 50
3.4.2 Quadratic Line Element with Three Nodes: Centroidal Coordinate 52
3.4.3 Linear Triangular Element with Three Nodes: Global Coordinate 53
3.4.4 Quadratic Triangular Element with Six Nodes 55
3.4.5 Linear Quadrilateral Element with Four Nodes: Centroidal Coordinate 57
3.5 Isoparametric Elements: Curved Boundaries 59
3.6 Numerical Evaluation of Integrals 63
3.6.1 Line Integrals 63
3.6.2 Triangular Area Integrals 67
3.6.3 Quadrilateral Area Integrals 69
3.7 Problems 72

4 ANSYS Preprocessor 75
4.1 Fundamentals of Modeling 75
4.2 Modeling Operations 75
4.2.1 Title 76
4.2.2 Elements 76
4.2.3 Real Constants 82
4.2.4 Material Properties 84
4.2.5 Element Attributes 86
4.2.6 Interaction with the Graphics Window: Picking Entities 87
4.2.7 Coordinate Systems 90

4.2.8 Working Plane 92
4.3 Solid Modeling 95
4.3.1 Bottom-up Approach: Entities 96
4.3.2 Top-Down Approach: Primitives 102
4.4 Boolean Operators 107
4.4.1 Adding 108
4.4.2 Subtracting 108
4.4.3 Overlap 110
4.4.4 Gluing 110
4.4.5 Dividing 111
4.5 Additional Operations 113
4.5.1 Extrusion 113
4.5.2 Moving and Copying 117
4.5.3 Keeping/Deleting Original Entities 119
4.5.4 Listing Entities 120
4.5.5 Deleting Entities 120
4.6 Viewing a Model 121
4.6.1 Plotting: PanZoom, and Rotate Functions 121
4.6.2 Plotting/Listing Entities 123
4.6.3 Numbers in the Graphics Window 124
4.7 Meshing 125
4.7.1 Automatic Meshing 125
4.7.2 Manipulation of the Mesh 132
4.8 Selecting and Components 133
4.8.1 Selecting Operations 133
4.8.2 Components 136

5 ANSYS Solution and Postprocessing 139
5.1 Overview 139
5.2 Solution 140
5.2.1 Analysis Options/Solution Controls 141
5.2.2 Boundary Conditions 143
5.2.3 Initial Conditions 144
5.2.4 Body Loads 144
5.2.5 Solution in Single and Multiple Load Steps 145
5.2.6 Failure to Obtain Solution 148
5.3 Postprocessing 149
5.3.1 General Postprocessor 150
5.3.2 Time History Postprocessor 150
5.3.3 Read Results 151
5.3.4 Plot Results 153
5.3.5 Element Tables 156
5.3.6 List Results 159
5.4 Example: One-dimensional Transient Heat Transfer 159

6 Finite Element Equations 177
6.1 Method of Weighted Residuals 177
6.1.1 Example: One-Dimensional Differential Equation with Line Elements 179
6.1.2 Example: Two-Dimensional Differential Equation with Linear Triangular Elements 187
6.1.3 Example: Two-Dimensional Differential Equation with Linear Quadrilateral Elements 203
6.2 Principle of Minimum Potential Energy 221
6.2.1 Example: One-Dimensional Analysis with Line Elements 228
6.2.2 Two-Dimensional Structural Analysis 234
6.3 Problems 272

7 Use of Commands in ANSYS 281
7.1 Basic ANSYS Commands 281
7.1.1 Operators and Functions 285
7.1.2 Defining Parameters 286
7.2 A Typical Input File 290
7.3 Selecting Operations 292
7.4 Extracting Information from ANSYS 297
7.5 Programming with ANSYS 300
7.5.1 DO Loops 300
7.5.2 IF Statements 302
7.5.3 /OUTPUT and *VWRITE Commands 304
7.6 Macro Files 306
7.7 Useful Resources 308
7.7.1 Using the Log File for Programming 308
7.7.2 Using the Verification Problems for Programming 310

8 Linear Structural Analysis 313
8.1 Static Analysis 313
8.1.1 Trusses 313
8.1.2 Beams 320
8.1.3 Three-Dimensional Problems 325
8.1.4 Two-Dimensional Idealizations 329
8.1.5 Plates and Shells 355
8.2 Linear Buckling Analysis 383
8.3 Thermomechanical Analysis 392
8.4 Fracture Mechanics Analysis 400
8.5 Dynamic Analysis 411
8.5.1 Modal Analysis 413
8.5.2 Harmonic Analysis 423
8.5.3 Transient Analysis 437

9 Linear Analysis of Field Problems 455
9.1 Heat Transfer Problems 455
9.1.1 Steady-state Analysis 456
9.1.2 Transient Analysis 497
9.1.3 Radiation Analysis 519
9.2 Moisture Diffusion 525

10 Nonlinear Structural Analysis 539
10.1 Geometric Nonlinearity 542
10.1.1 Large Deformation Analysis of a Plate 543
10.1.2 Post-buckling Analysis of a Plate with a Hole 546
10.2 Material Nonlinearity 551
10.2.1 Plastic Deformation of an Aluminum Sphere 552
10.2.2 Plastic Deformation of an Aluminum Cylinder 555
10.2.3 Stress Analysis of a Reinforced Viscoelastic Cylinder 562
10.2.4 Viscoplasticity Analysis of a Eutectic Solder Cylinder 566
10.2.5 Combined Plasticity and Creep 572
10.3 Contact 579
10.3.1 Contact Analysis of a Block Dropping on a Beam 581
10.3.2 Simulation of a Nano-Indentation Test 587

11 Advanced Topics in ANSYS 595
11.1 Coupled Degrees of Freedom 595
11.2 Constraint Equations 597
11.3 Submodeling 603
11.4 Substructuring: Superelements 609
11.4.1 Generation Pass 612
11.4.2 Use Pass 618
11.4.3 Expansion Pass 620
11.5 Interacting with External Files 621
11.5.1 Reading an Input File 622
11.5.2 Writing Data to External ASCII Files 622
11.5.3 Executing an External File 625
11.5.4 Modifying ANSYS Results 627
11.6 Modifying the ANSYS GUI 628
11.6.1 GUI Development Demonstration 635
11.6.2 GUI Modification for Obtaining a Random Load Profile 643
11.6.3 Function Block for Selecting Elements Using a Pick Menu 648

Erratum to: The Finite Element Method and Applications in Engineering Using ANSYS® - Supplemental Materials E1

References 651

Index 653

List of Problems Solved

ANSYS Solution of a Two-dimensional Differential Equation with Linear Triangular Elements 199
ANSYS Solution of a Two-dimensional Differential Equation with Linear Quadrilateral Elements 217
Plane Stress Analysis with Linear Triangular Elements 248
Plane Stress Analysis with Linear Quadrilateral Isoparametric Elements 268
Elongation of a Bar Under Its Own Weight Using Truss Elements 314
Analysis of a Truss Structure with Symmetry 317
Analysis of a Slit Ring 321
Elongation of a Bar Under Its Own Weight Using 3-D Elements 325
Plane Stress Analysis of a Plate with a Circular Hole 329
Plane Stress Analysis of a Composite Plate Under Axial Tension 337
Plane Strain Analysis of a Bi-material Cylindrical Pressure Vessel Under Internal Pressure 342
Deformation of a Bar Due to Its Own Weight Using 2-D Axisymmetric Elements 348
Analysis of a Circular Plate Pushed Down by a Piston Head Using 2-D Axisymmetric Elements 350
Static Analysis of a Bracket Using Shell Elements 356
Analysis of a Circular Plate Pushed Down by a Piston Head Using Solid Brick and Shell Elements 365
Analysis of an Axisymmetric Shell with Internal Pressure Using Shell Elements 372
Analysis of a Layered Composite Plate Using Shell Elements 378
Linear Buckling Analysis of a Plate 382
Thermomechanical Analysis of an Electronic Package 392
Fracture Mechanics Analysis of a Strip with an Inclined Edge Crack 400
Modal Analysis of a Bracket 413
Vibration Analysis of an Automobile Suspension System 416
Harmonic Analysis of a Bracket 423
Harmonic Analysis of a Guitar String 431
Dynamic Analysis of a Bracket 439

Impact Loading on a Beam 443
Dynamic Analysis of a 4-bar Linkage 449
Heat Transfer Analysis of a Tank/Pipe Assembly 456
Heat Transfer Analysis of a Window Assembly 477
Transient Thermomechanical Analysis of an Electronic Package 498
Transient Thermomechanical Analysis of a Welded Joint 509
Radiation Heat Transfer Analysis of a Conical Fin 519
Moisture Diffusion Analysis of an Electronic Package 525
Large Deformation Analysis of a Plate 543
Postbuckling Analysis of a Plate with a Hole 546
Plastic Deformation of an Aluminum Sphere 552
Plastic Deformation of an Aluminum Cylinder 555
Stress Analysis of a Reinforced Viscoelastic Cylinder 562
Viscoplasticity Analysis of a Eutectic Solder Cylinder 566
Combined Plasticity and Creep Analysis of a Eutectic Solder Cylinder 572
Contact Analysis of a Block Dropping on a Beam 581
Simulation of a Nano-indentation Test 587
Analysis of a Sandwich Panel Using Constraint Equations 597
Submodeling Analysis of a Square Plate with a Circular Hole 603
Substructuring Analysis of an Electronic Package 609
GUI Development Demonstration 635