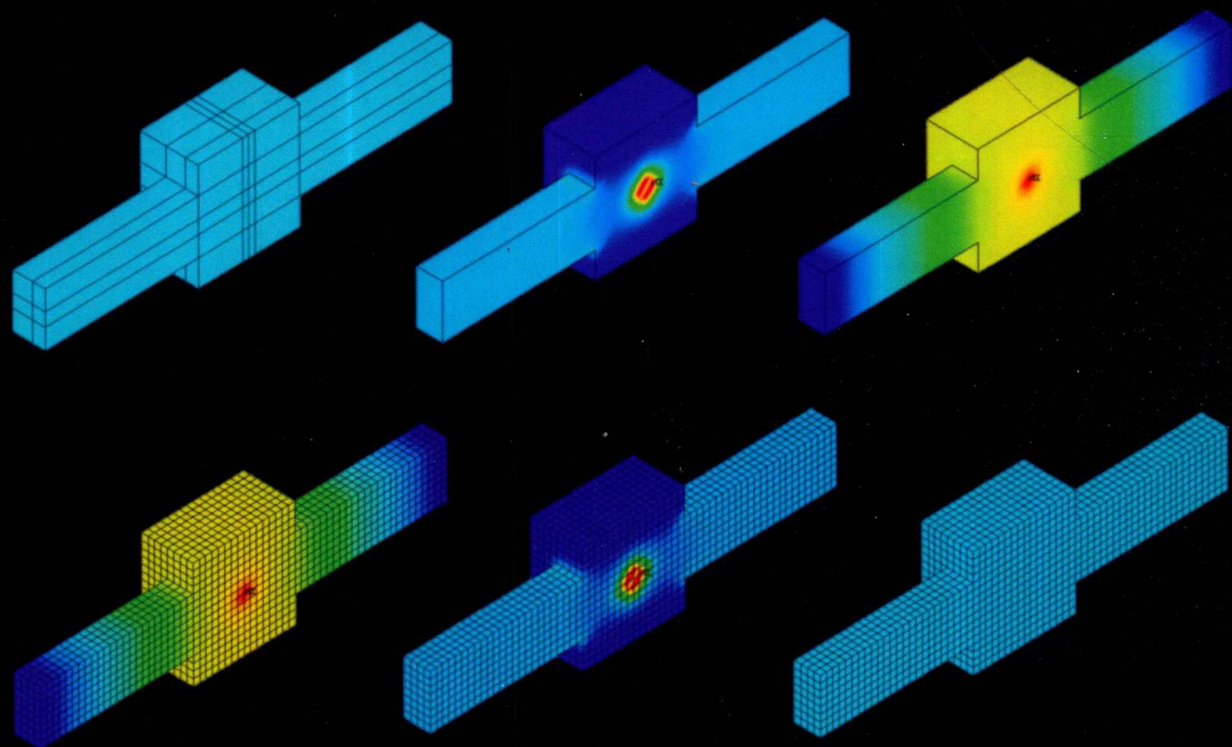
ANSYS Mechanical APDL for Finite Element Analysis



Mary Kathryn Thompson
John M. Thompson

BH

ANSYS Mechanical APDL for Finite Element Analysis

Mary Kathryn Thompson, PhD

John M. Thompson, PhD, PE

ELSEVIER

Butterworth-Heinemann
An imprint of Elsevier

Butterworth-Heinemann is an imprint of Elsevier
The Boulevard, Langford Lane, Kidlington, Oxford OX5 1GB, United Kingdom
50 Hampshire Street, 5th Floor, Cambridge, MA 02139, United States

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

Library of Congress Cataloging-in-Publication Data
A catalog record for this book is available from the Library of Congress

ISBN: 978-0-12-812981-4

For Information on all Butterworth-Heinemann publications
visit our website at https://www.elsevier.com/books-and-journals

Publisher: Joe Hayton
Acquisition Editor: Brian Guerin
Editorial Project Manager: Katie Chan
Production Project Manager: Kiruthika Govindaraju
Cover Designer: Mary Kathryn Thompson, John M. Thompson, Mark Rogers

Typeset by MPS Limited, Chennai, India

ANSYS Mechanical APDL for Finite Element Analysis

To Veronica

Mother, Wife, Inspiration

Preface

Commercial finite element programs provide a powerful and extensive collection of tools for the design, analysis, and optimization of complex engineering systems. Unfortunately, learning to use finite element software can be a difficult and time-consuming process. This book was written to reduce the learning curve associated with ANSYS Mechanical APDL and to prepare you to use the program and its documentation as quickly and painlessly as possible.

This book was written using the Academic Research version of ANSYS Mechanical APDL 17.2. Since ANSYS Mechanical APDL was designed to be backward compatible, you should be able to use the book with future versions of the program as well.

We have marked information in the book that we believe is likely to become out of date quickly, such as specific product names and the number of reference manuals for a given product, with a superscript asterisk (*) for your reference. An appendix has also been included to help you to identify specific information in the program documentation, even if the documentation numbering changes. However, the authors of this book are not involved with changes that may be made in the future to the ANSYS Mechanical APDL software or the related documentation. Therefore the asterisks and appendix are for guidance only and may not identify all future changes in such software or documentation.

Each chapter in the book includes suggested readings from the ANSYS Mechanical APDL user manuals. Most chapters are also accompanied by one or more hands-on exercises. This book does not duplicate or replace the information in the documentation. Instead, it is intended to help you to understand and use the documentation. Thus the reading assignments in the ANSYS user manuals are strongly recommended. To help you develop comfort and confidence as an independent analyst, each exercise introduces new skills and increases the complexity of the models while gradually decreasing the amount of guidance given. We recommend completing all of the exercises, even if they are not immediately relevant to your work.

This book can be used for self-study or as part of a formal course in finite element applications. It is suitable for professional engineers and engineering students at all levels. When you have finished the book, you should be able to use the present ANSYS documentation to build, solve, and postprocess sophisticated finite element models on your own.

The world is full of interesting and important problems. This book gives you some of the tools that you need to help solve them. Good luck!

Mary Kathryn Thompson and John M. Thompson
Pittsburgh, PA, USA

Acknowledgments

This book was originally developed as the course text for a graduate level IAP course at the Massachusetts Institute of Technology (MIT). The goal of that course was to prepare students to perform research quality finite element analyses with ANSYS in 5 days or less. We began writing in 2002 and began teaching with the book in 2004. In 2008 the book and the course moved to Korea. In 2012 the book moved to Denmark. And in 2016 it moved home again to the United States. Over the years, we have received a tremendous amount of support from wonderful people all over the world. We would like to acknowledge some of them here.

The software licenses for the MIT IAP course and for the development of this book were provided through an ANSYS Academic Partnership. We are grateful for the generous support that we have received from ANSYS, Inc. over the years. We would especially like to acknowledge the late Mr. Jerry Bittner, former Director of ANSYS Global Technical Services, who started this journey with us, and Dr. Paul Lethbridge, Senior Manager for Academic and Start Up Programs at ANSYS, who saw it through to the end. We would also like to thank Sheryl Ackerman, Cordell Blackshere, Vishal Ganore, Helen Renshaw, and the rest of the incredible team, past and present, at ANSYS Worldwide Headquarters in Canonsburg, PA, USA.

We would like to thank Prof. Alexander H. Slocum, Prof. Rohan Abeyaratne, Prof. David M. Parks, Prof. Lallit Anand, Prof. Thomas Peacock, and the MIT Department of Mechanical Engineering; Prof. Heekyung Park, Prof. Chung-Bang Yun, and the KAIST Department of Civil and Environmental Engineering; and Prof. Hans Nørgaard Hansen, Prof. Leonardo De Chiffre, and the DTU Department of Mechanical Engineering for their encouragement and support of this project as it has traveled around the world.

We would like to acknowledge Dr. Veronica V. Thompson for her tremendous assistance in editing this book. She has helped us to become the writers we are.

We would like to thank Anne M. Thompson and Will C. Lauer for all of their logistical and moral support over the years. We could not have done it without them.

We would like to thank Courtney S. Bermack, Jeffrey Chambers, Chad Foster, Marissa Jacovich, Christina Laskowski, Michael Mischkot, A. Zachary Trimble, and Antonio Vicente for their help in testing and improving various versions of this book. Their help and feedback is greatly appreciated.

We would like to thank our wonderful publication team at Elsevier, especially Brian Guerin, Katie K. Chan, and Kiruthika Govindaraju who made this process a pleasure.

Finally, we would like to thank Prof. Nam P. Suh, Prof. Donald A. Norman, Prof. Sami Kara, and Prof. Guan Heng Yeoh. They have all provided crucial advice and opportunities at critical times. We are glad to have traveled with them on this journey.

Contents

Preface		**xv**
Acknowledgments		**xvii**
1.	**Introduction to ANSYS and Finite Element Modeling**	**1**
1.1	What Is the Finite Element Method?	1
1.2	Why Use the Finite Element Method?	2
1.3	Basic Procedure for Finite Element Analysis	2
1.4	Engineering Software—Not an Engineer	3
1.5	A Brief History of ANSYS and Finite Element Analysis	3
1.5.1	The Development of NASTRAN	4
1.5.2	The Development of ANSYS	4
1.5.3	The Evolution of ANSYS	4
1.6	ANSYS Today	5
1.7	ANSYS Licensing	6
1.8	Functionality and Features of the ANSYS Mechanical APDL Family	6
1.8.1	Can ANSYS...?	6
1.8.2	Steady-State and Time-Dependent Analyses	6
1.8.3	Physics Capabilities	7
1.8.4	Special Features	7
1.9	ANSYS: Backward Compatibility and Legacy Code	8
2.	**Interacting with ANSYS**	**11**
2.1	ANSYS Simulation Environments	12
2.2	Communicating with ANSYS	13
2.2.1	ANSYS Commands	13
2.2.2	The Graphical User Interface	14
2.2.3	The GUI Command Prompt	16
2.2.4	Input Files and Batch Files	17
2.3	How ANSYS Communicates with You	18
2.3.1	INFO Level Feedback	18
2.3.2	NOTE Level Feedback	18
2.3.3	WARNING Level Feedback	19
2.3.4	ERROR Level Feedback	19
2.3.5	FATAL Level Feedback	19
2.4	ANSYS Program Structure	19
2.4.1	Levels and Processors	19
2.4.2	The ANSYS Database	20
2.4.3	Types of Commands and Their Locations	21
2.5	ANSYS File Structure	22
2.5.1	The Database File	23
2.5.2	The Log File	23
2.5.3	The Lock File	23
2.5.4	The Error File	24
2.5.5	The Output File	24
2.5.6	The Results File	24

2.6 Saving Files and Results in ANSYS 24
2.6.1 Saving Database Files 24
2.6.2 Archiving Models 25
2.6.3 Rerunning Log Files 25
2.6.4 Creating Input and Batch Files 25
2.7 Where is the Undo Button? 26
2.8 How Do You Specify Units? 26
2.9 Where to Find Help: The ANSYS Documentation 27
2.9.1 Reference Manuals 28
2.9.2 Programmer's Manuals 28
2.9.3 Examples Manuals 28
2.9.4 Analysis Guides 29
2.9.5 The Feature Archive 29
2.9.6 Additional Documentation 29
2.10 Where to Get Extra Help: ANSYS Technical Support 30

Exercise 2-1 Static Axial Loading of a Notched Plate in Tension 31

3. Creating and Importing Geometry 47
3.1 Considerations for Model Geometry 47
3.1.1 Choosing Direct Generation or Solid Modeling 47
3.1.2 Choosing Whether to Create or Import Solid Model Geometry 48
3.1.3 Choosing the Dimensionality of the Model 48
3.1.4 Choosing How Much Detail to Include 50
3.2 Creating Model Geometry 50
3.2.1 Direct Generation of Nodes and Elements 50
3.2.2 Creating Model Geometry from the Bottom-Up 52
3.2.3 Creating Model Geometry from the Top Down 54
3.3 Boolean Operations 54
3.3.1 Boolean Options 55
3.3.2 Number Merging 56
3.3.3 Numbering in Boolean Operations 58
3.3.4 Boolean Operations: Model First, Mesh Second 58
3.3.5 Boolean Operation Errors 58
3.4 Deleting Solid Model Geometry 58
3.5 Importing Solid Model Geometry 58
3.5.1 Importing Solid Models Using IGES Files 59
3.5.2 Importing Solid Models Using Connection Products 59
3.5.3 Importing CAD Using ANSYS Workbench and DesignModeler 59
3.6 Coordinate Systems 59
3.6.1 Global Coordinate Systems 59
3.6.2 Local Coordinate Systems 60
3.6.3 The Display Coordinate System 60
3.7 The Working Plane 61
3.8 Solid Model Viewing 62
3.8.1 List 62
3.8.2 Plot 63
3.8.3 PlotCtrls 63

Exercise 3-1 Bottom-Up Solid Modeling of a Plate With a Central Hole Using Quarter Symmetry 67

Exercise 3-2 Top-Down Solid Modeling of a Pipe Flange Using Symmetry 77

Exercise 3-3 Structural Analysis of a Simple Warren Truss Using Direct Generation 95

4. Elements and Element Input	**107**
4.1 Element Classification in ANSYS	108
4.1.1 Current-Technology Elements	108
4.1.2 GUI-Inaccessible Elements	108
4.1.3 Legacy Elements	108
4.1.4 Undocumented Elements	109
4.1.5 Superelements	109
4.1.6 User Elements	109
4.2 The ANSYS Element Library	109
4.3 Element Properties	110
4.3.1 Element Names	111
4.3.2 Element Shapes	112
4.3.3 Number of Nodes	112
4.3.4 Degenerate Shapes	113
4.3.5 Element Shape Functions and Extra Displacement Shapes	114
4.3.6 Degrees of Freedom	115
4.3.7 Real Constants	115
4.3.8 Key Options	115
4.3.9 Required and Permitted Material Properties	115
4.3.10 Permitted Loads	116
4.3.11 Special Features	116
4.4 ANSYS Element Families	116
4.5 Product Codes and Product Restrictions	118
4.6 Choosing an Element	120
4.6.1 Bottom-Up Element Selection: Commonly Used Elements	120
4.6.2 Top-Down Element Selection: Process of Elimination	120
4.7 Defining Element Types	121
4.8 Deleting Element Types	122
4.9 Defining Real Constants	122
4.10 Defining Sections	122
Exercise 4-1 Modeling a Simple 1D Cantilever Beam Using Beam Elements	**123**
Exercise 4-2 Modeling a Simple 2D Cantilever Beam Using PLANE Elements	**131**
Exercise 4-3 Modeling a Simple 3D Cantilever Beam Using SOLID Elements	**139**
5. Defining Material Properties	**147**
5.1 What are Material Models?	147
5.2 Material Models in ANSYS	148
5.2.1 Material Model Name	148
5.2.2 Linearity	148
5.2.3 Material Property and Material Model Labels	148
5.2.4 Material Property Values	149
5.2.5 Spatial Dependence	149
5.2.6 Temperature Dependence	150
5.2.7 Supported Material Model Combinations	150
5.2.8 Supported Elements	150
5.2.9 Product Restrictions	150
5.3 Defining Material Properties	151
5.3.1 Inputting Linear Material Property Values Using the GUI	151
5.3.2 Inputting Linear Material Property Values Using Commands	152
5.3.3 Inputting Nonlinear Material Property Values Using the GUI	152

5.3.4 Inputting Nonlinear Material Property Values Using Commands 153
5.3.5 Inputting Nonlinear Material Properties Curves Using the GUI 154
5.3.6 Inputting Nonlinear Material Property Curves Using Commands 155
5.3.7 Inputting Temperature-Dependent Material Property Data Using the GUI 155
5.3.8 Inputting Linear Temperature-Dependent Material Property Data Using Commands: Polynomial Equations 156
5.3.9 Specifying Temperature Dependence Using Commands: Temperature Tables 156
5.3.10 How the GUI Specifies Linear Material Properties Using Commands 157
5.3.11 Inputting Nonlinear Temperature-Dependent Material Property Data Using Commands 157
5.3.12 GUI Inaccessible Materials 158
5.3.13 User Materials 158
5.4 Choosing Which Material Properties to Define 158
5.5 Finding Material Property Data 159
5.6 Potential Pitfalls Associated with Material Property Evaluation in ANSYS 159
5.6.1 Too Few Material Properties 159
5.6.2 Too Many Material Properties 159
5.6.3 Insufficient Number of Points on a Material Data Curve 160
5.7 Saving Material Properties 160
5.7.1 Exporting Material Properties as ANSYS Material Library Files 160
5.7.2 Saving Material Properties as Commands 161

Exercise 5-1 Temperature-Dependent Plasticity Analysis of a Plate with a Central Hole 163

6. Meshing 181
6.1 Meshing Overview 181
6.2 Element Attributes 182
6.2.1 Setting Global Element Attributes 183
6.2.2 Setting Local Element Attributes 184
6.2.3 Modifying Element Attributes 184
6.3 Mesh Controls 185
6.3.1 Free Versus Mapped Meshing 185
6.3.2 Element Sizing 186
6.3.3 Always Save Before Meshing 189
6.4 Generating a Mesh 189
6.4.1 The Mesh Tool 189
6.4.2 Meshing Commands 190
6.4.3 Meshing Order 190
6.5 Mapped Meshing 191
6.5.1 Map Meshing Using Boolean Operations 191
6.5.2 Map Meshing Using Concatenation 191
6.5.3 Map Meshing Areas By Corners 192
6.6 Copying and Extruding a Mesh 192
6.7 Defining the Quality of a Mesh 192
6.7.1 The Mesh Must Accurately Represent the System 193
6.7.2 The Mesh Must Generate Accurate Results 193
6.8 Determining the Quality of a Mesh 195
6.8.1 Evaluating Element Shapes with Element Shape Testing 195
6.8.2 Evaluating Mesh Density by Visual Inspection 196
6.8.3 Evaluating Mesh Density through Energy Error Estimation 197
6.8.4 Evaluating the Mesh Quality through Mesh Convergence 198

6.9 Modifying and Regenerating a Mesh 198
6.9.1 Refining a Mesh 198
6.9.2 Clearing a Mesh 199

Exercise 6-1 Determining the Mesh Convergence of a Heated Plate With a Central Hole 201

7. Selecting Entities 239
7.1 Specifying Entities in ANSYS 239
7.2 Selection Overview 240
7.2.1 Type of Entity 240
7.2.2 Selection Methods 240
7.2.3 Set Operations 241
7.3 Selecting Entities in ANSYS 242
7.3.1 The Select Menu 242
7.3.2 Select Commands 243
7.3.3 Examples Using Select Commands 244
7.3.4 Saving Selection Sets Using Components and Assemblies 246
7.4 The Picker 247
7.5 Picker Commands 248
7.5.1 The FLST Helper Command 249
7.5.2 The FITEM Helper Command 249
7.5.3 Picker Command Blocks for Selecting Operations 249
7.5.4 Picker Command Blocks for Picking Operations 250

8. Solution 253
8.1 Defining "Solution" 253
8.2 Boundary Condition Overview 254
8.2.1 Constraints 254
8.2.2 Loads 254
8.3 Boundary Conditions in ANSYS 254
8.3.1 Boundary Conditions in ANSYS—Organized by Physics 255
8.3.2 Boundary Conditions in ANSYS—Organized by Entity 255
8.3.3 Boundary Conditions in ANSYS—Organized by Interaction 255
8.4 Applying Boundary Conditions 256
8.4.1 Deleting Boundary Conditions 257
8.4.2 Confirming Boundary Conditions in ANSYS 257
8.5 Potential Pitfalls Associated with Applying Boundary Conditions in ANSYS 258
8.5.1 No DOF to Constrain 258
8.5.2 Too Few Constraints 259
8.5.3 Too Many Constraints 259
8.5.4 Mixing Loads and Constraints 260
8.5.5 Repeating Boundary Conditions 261
8.5.6 Mixing Solid Model and Finite Element Model Boundary Conditions 262
8.6 Initial Conditions in ANSYS 263
8.7 Solution Options 263
8.7.1 Analysis Type 264
8.7.2 Small vs Large Displacement 264
8.7.3 Load Steps 265
8.7.4 Substeps 265
8.7.5 Time Steps 266
8.7.6 Auto Time Stepping 267

8.7.7 Output Options 267
8.7.8 Solvers 267
8.8 Initiating a Solution 268
8.8.1 Confirming the Solution Options 268
8.8.2 Solution Checking 269
8.9 During Solution 270
8.9.1 Feedback During Solution 270
8.9.2 Solution Failure Modes 271
8.9.3 Terminating a Running Job 272
8.10 After Solution 273
8.10.1 Change the Jobname 273
8.10.2 Restart the Analysis 273

Exercise 8-1 Time Varying Heat Conduction Through a Composite Wall 275

9. Postprocessing 295
9.1 Postprocessing Overview 295
9.2 Types of Results 296
9.3 Available Results 297
9.4 Accessing Results From the Output File 299
9.5 Accessing Results From the Results File 301
9.5.1 Accessing the Results File Through the General Postprocessor 302
9.5.2 Accessing Results from the Results File Through the Time History Postprocessor 303
9.5.3 Accessing Results via Component Name or Sequence Number 303
9.6 Results Coordinate Systems 305
9.6.1 The Nodal Coordinate System 305
9.6.2 Element Coordinate Systems 305
9.6.3 The Results (Display) Coordinate System 305
9.7 Full Graphics vs PowerGraphics 307
9.8 Displaying and Viewing Results 308
9.8.1 Listing Results 308
9.8.2 Plotting Results 311
9.8.3 Animating Results 315
9.8.4 Graphing Results 315
9.9 Postprocessing With Load Case Combinations 316
9.10 Saving Postprocessing Graphics and Information 317
9.11 Model Verification and Validation 317

Exercise 9-1 Postprocessing an Axisymmetric Cylindrical Pressure Vessel Using Element Tables 319

Exercise 9-2 Postprocessing a 3D Thermal Model With Geometric Discontinuities Using Power Graphics 337

Exercise 9-3 Postprocessing a Cylindrical Structural Shell Using PowerGraphics, Results Coordinate Systems, and Load Case Combinations 355

10. Input Files 371
10.1 Approaches to Writing Input Files 371
10.1.1 The Direct Method for Preparing an Input File 371
10.1.2 The Sequential Method for Preparing an Input File 372
10.1.3 The Concurrent Method for Preparing an Input File 372

10.2 Tools for Writing and Debugging Input Files 373
10.2.1 Plain Text Editor 373
10.2.2 Mechanical APDL Command Dictionary 373
10.2.3 Sketching Aids 374
10.3 Accessing the Log File 375
10.3.1 Open the Session Log File From the Working Directory 375
10.3.2 List the Session Log File From Within ANSYS 375
10.3.3 Export and Open the Database Log File 375
10.4 Common Features of GUI-Generated Log Files 376
10.4.1 The Header Block 376
10.4.2 Release and Time Stamps 376
10.4.3 LGWRITE Commands 376
10.4.4 Material Property Blocks 377
10.4.5 Picker Blocks 377
10.4.6 Nonessential Commands 378
10.4.7 Extra Commands 378
10.4.8 Repeated Commands 379
10.4.9 Reversed Commands 380
10.4.10 Commands That Generate Warnings or Errors 380
10.4.11 Extra Spaces 380
10.4.12 Extra Syntax 380
10.5 Guidelines for the Sequential Method 381
10.5.1 Procedure for Editing a Log File 381
10.5.2 Procedure for Extracting Commands From a Log File 382
10.6 Debugging an Input File 383
10.6.1 Types of Input File Errors 383
10.6.2 End-of-File Commands 383
10.6.3 The Debugging Process 384
10.7 Documenting Your Work 384
10.7.1 Documenting Modeling Assumptions and Decisions 384
10.7.2 Commenting an Input File 385

Exercise 10-1 Using the Sequential Method to Create an Input File for 1D Steady-State Conduction Through a Steel Clad Copper Pan 387

Exercise 10-2 Using the Concurrent Method to Modify an Input File for Steady-State Conduction Through a Cladded Plate 407

Exercise 10-3 Using the Direct Method to Create a Batch File for Steady-State Conduction Through a Cladded Plate With Varying Surface Temperatures 419

Appendix: Chapter and Section Numbering for Selected ANSYS Mechanical APDL 17.2 Documentation 437

Index 441

Chapter

1

Introduction to ANSYS and Finite Element Modeling

Suggested Reading Assignments:
None

CHAPTER OUTLINE

1.1 What Is the Finite Element Method?
1.2 Why Use the Finite Element Method?
1.3 Basic Procedure for Finite Element Analysis
1.4 Engineering Software—Not an Engineer
1.5 A Brief History of ANSYS and Finite Element Analysis
1.6 ANSYS Today*
1.7 ANSYS Licensing
1.8 Functionality and Features of the ANSYS Mechanical APDL Family
1.9 ANSYS: Backward Compatibility and Legacy Code

This chapter provides an introduction to finite element analysis and the ANSYS Mechanical APDL family of software. It begins with an overview of the finite element method, its benefits, and its limitations. It summarizes the current ANSYS Mechanical APDL products and program capabilities. Finally, it describes the program's evolution and how that influences the use of ANSYS, Inc. products.

1.1. What Is the Finite Element Method?

The finite element method (FEM) is a mathematical technique for setting up and solving systems of partial differential (or integral) equations. In engineering, the finite element method is used to divide a system whose behavior cannot be predicted using closed form equations into small pieces, or elements, whose solution is known or can be approximated. The finite element method requires the system geometry to be defined by a number of points in space called nodes. Each node has a set of degrees of freedom (temperature, displacements, etc.) that can vary based on the inputs to the system. These nodes are connected by elements that define the mathematical interactions of the degrees of freedom (DOFs). For some elements, such as beams, the closed form solution is known. For other elements, such as continuum elements, the interaction among

ANSYS Mechanical APDL for Finite Element Analysis.
DOI: http://dx.doi.org/10.1016/B978-0-12-812981-4.00001-0

the degrees of freedom is estimated by a numerical integration over the element. All individual elements in the model are combined to create a set of equations that represent the system to be analyzed. Finally, these equations are solved to reveal useful information about the behavior of the system.

Just as a regular polygon approaches a perfect circle as the number of sides approaches infinity, a finite element model approaches a perfect representation of the system as the number of elements becomes infinite. Since it is impossible to divide the system into an infinite number of elements, the finite element method produces the exact solution to an approximation of the problem that you want to solve. When the number of elements becomes sufficiently large, the approximation becomes good enough to use for engineering analysis. However, this may increase the number of equations to be solved beyond the point where it is practical or desirable to solve them by hand. For this reason, the finite element method is associated with computer programs that set up, solve, and visualize the solutions of these large sets of equations for you.

1.2. Why Use the Finite Element Method?

The cost, in terms of the manpower and computer resources, required to set up and solve a finite element model for a simple problem like a cantilever beam is very high compared to the benefit. Simple problems can—and should—be solved with simple methods (or obtained from engineering handbooks). But not all problems are simple. For example, if a bridge is built using a simple truss supported by two piers, the deflections and stresses in the bridge can be found using information taught in an introductory statics and strength of materials class. But as the complexity of the truss increases, solving this problem using the engineering fundamentals becomes more difficult, leaving the analyst with long hours of error-prone calculations. As system complexity continues to increase, closed-form analysis rapidly becomes impossible. The real benefit of finite element analysis lies in the ability to solve arbitrarily complex problems for which analytical solutions are not available or which would be prohibitively time consuming and expensive to solve by hand.

1.3. Basic Procedure for Finite Element Analysis

There are 10 basic steps in any finite element analysis. First, the solid model geometry is created, the element type(s) and material properties are defined, and the solid model geometry is meshed to create the finite element model. In ANSYS, these steps are performed in the Preprocessor (PREP7). Next, loads and constraints are applied, solution options are defined, and the problem is solved. These steps are performed in the Solution processor (SOL). After the solution is ready, the results are plotted, viewed, and exported in one of the postprocessors (POST1 or POST26). Finally, the results are compared to first-order estimates, closed-form solutions, mathematical models, or experimental results to ensure that the output of the program is reasonable and as expected. (Processors will be addressed in more detail in chapter 2.)

/PREP7

1. Define the Solid Model Geometry
2. Select the Element Types
3. Define the Material Properties
4. Mesh

/SOLUTION

5. Define the Boundary Conditions
6. Define the Loads
7. Set the Solution Options
8. Solve

/POST1 or /POST26

9. Plot, View, and Export the Results
10. Compare and Verify the Results

It is sometimes possible to omit one or more steps. For example, the default solution options are often sufficient for a simple analysis. It is possible to perform some steps out of order. For example, the element types and material properties can be defined in either order. Similarly, the loads and boundary conditions can be defined in either order. It is occasionally necessary to perform these steps out of order. For example, solid model geometry is not required for a finite element analysis. When the nodes and elements are generated directly, the element type(s) must be specified before the geometry can be created. Finally, complicated analyses may involve multiple trips through one or more processors.

For simplicity, this 10-step procedure will be used in this book whenever possible.

1.4. Engineering Software—Not an Engineer

As with all computer programs, the quality of your results will depend on the quality of your model. This includes the accuracy of the material properties, the appropriateness of the material models, how closely the simulated geometry and loads match the actual geometry and loads, and the validity of the simplifications and assumptions made. Simply put, Garbage In = Garbage Out. Finite element software programs can be thought of as very sophisticated calculators that help you to analyze engineering systems that could not otherwise be evaluated. They integrate the section properties of the system with the material properties to generate the equations to be solved. They convert the applied loads to the appropriate forms and apply them to the specified DOFs. They solve the generated system of equations. And, they help you to visualize and understand the results. But a finite element program will not comment on the validity of any assumptions made in setting up the model as long as the laws of physics are not violated. It also will not ensure that you are using the correct laws of physics for a given problem. Any errors that the program reports will be associated with the use of the program, and not with the physical or analytical system. In addition, it will not provide any commentary on the quality or implications of the results. Finite element software is only a tool. In the end, you, and you alone, are responsible for determining whether or not the results of your finite element model can be used to make or justify engineering decisions.

1.5. A Brief History of ANSYS and Finite Element Analysis

The finite element method was first proposed in the early 1940s as a numerical technique for solving partial differential equations. At that time, a mesh of elements could be defined and the interaction of the elements could be used to create the system of equations to be solved. However, the system of equations still had to be solved by hand. This limitation rendered the finite element method an academic curiosity until the early 1960s when computers that could