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:

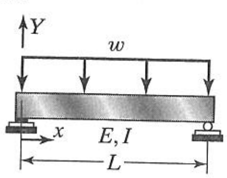
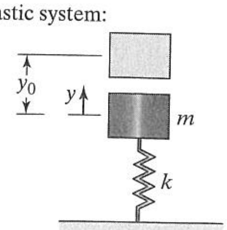
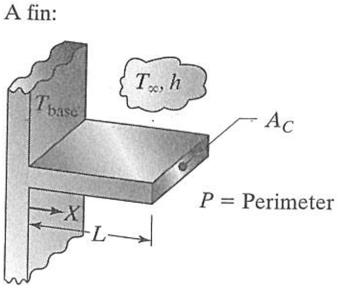
**1.1** Engineering Problems
**1.2** Numerical Methods
**1.3** A Brief History of the Finite Element Method and ANSYS
**1.4** Basic Steps in the Finite Element Method
**1.5** Direct Formulation
**1.6** Minimum Total Potential Energy Formulation
**1.7** Weighted Residual Formulations
**1.8** Verification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

Moaveni, Finite Element Analysis, 2nd ed, 2003

## 1.1 ENGINEERING PROBLEMS

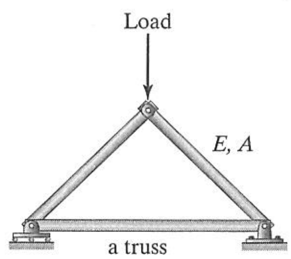
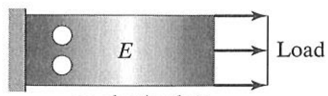
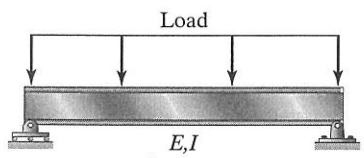
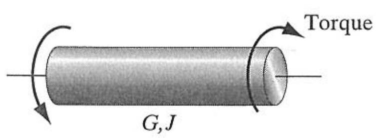
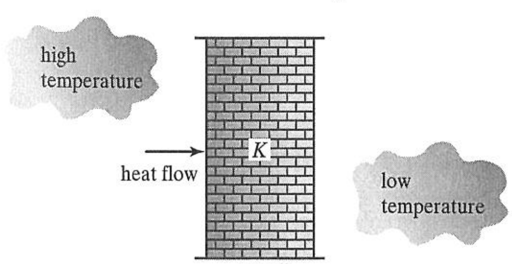
In general, engineering problems are mathematical models of physical situations. Mathematical models of many engineering problems are differential equations with a set of corresponding boundary and/or initial conditions. The differential equations are derived by applying the fundamental laws and principles 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 provide in-

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

| Problem Type | Governing Solution, 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 $X = 0, T = T_{base}$ as $L \rightarrow \infty$, $T = T_\infty$ | Temperature distribution along the fin as the function of $X$: $T = T_\infty + (T_{base} - T_\infty)e^{-\sqrt{\frac{hp}{kA_c}}X}$ |

formation 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, $E$; cross-sectional area, $A$ |
|   an elastic plate | modulus of elasticity, $E$ |
|   a beam | modulus of elasticity, $E$; second moment of area, $I$ |
|   a shaft | modulus of rigidity, $G$; 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*

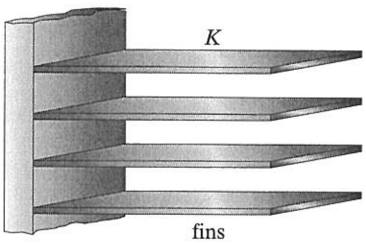
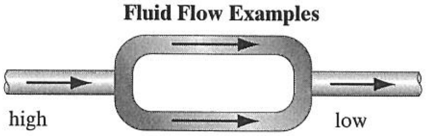
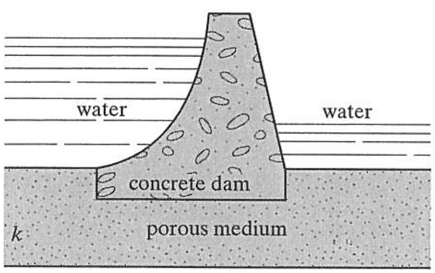
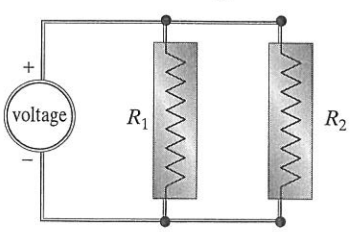
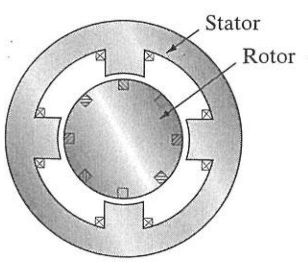
| Problem Type | Examples of Parameters That Characterize a System |
|---|---|
|   fins | thermal conductivity, $K$; Cross-Sectional Area, $A$ |
| **Fluid Flow Examples** | |
|   pipe networks | viscosity, $\mu$; pipe roughness, $e$; pipe diameter, $D$ |
|   a concrete dam | soil permeability, $k$ |
| **Electrical and Magnetism Problems** | |
|   electrical network | resistance, $R$ |
|   magnetic field of an electric motor | permeability, $\mu$ |

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.

## 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 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 over 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 well over 20 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:

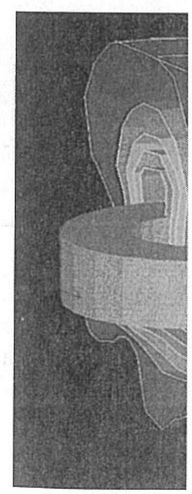
**Preprocessing Phase**

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 Exampl

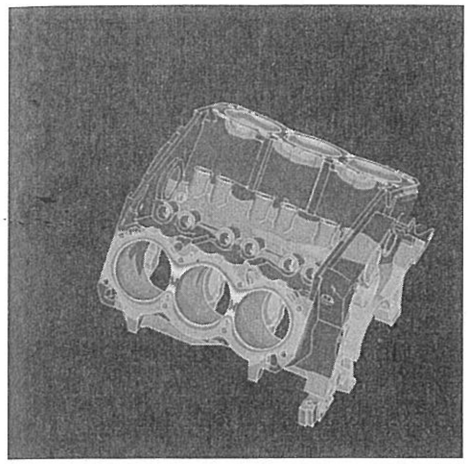
A V6 engine used in
analyzed using ANS
analyses were condu
Co. Ltd. (ADAPCO)
automobile manufac
performance. Conto
block are shown in t



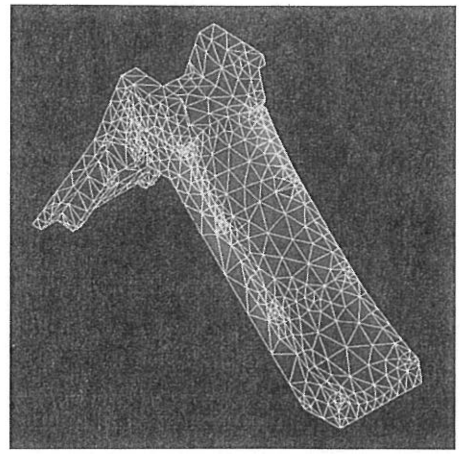
Electromagnetic cap
the use of both vecto
through a specialize
dimensional graphics
through infinite bou
this analysis of a batl
Isocontours are used
H-field.

*Photographs courtesy

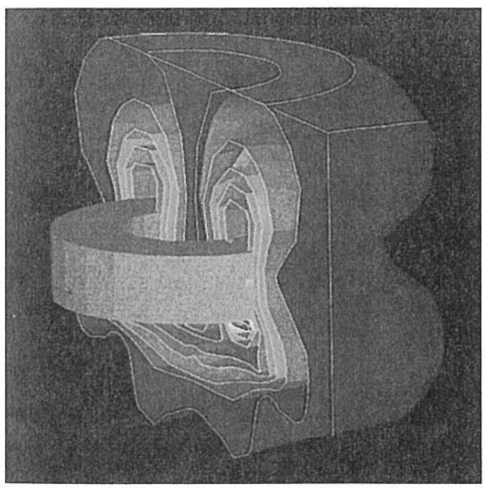
TABLE 1.4 Examples of the capabilities of ANSYS*



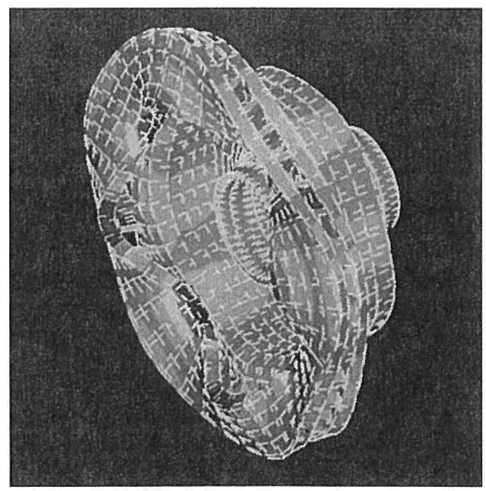
A V6 engine used in front-wheel-drive automobiles was analyzed using ANSYS heat transfer capabilities. The analyses were conducted by Analysis & Design Appl. Co. Ltd. (ADAPCO) on behalf of a major U.S. automobile manufacturer to improve product performance. Contours of thermal stress in the engine block 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, an approximate continuous function is assumed to represent the 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