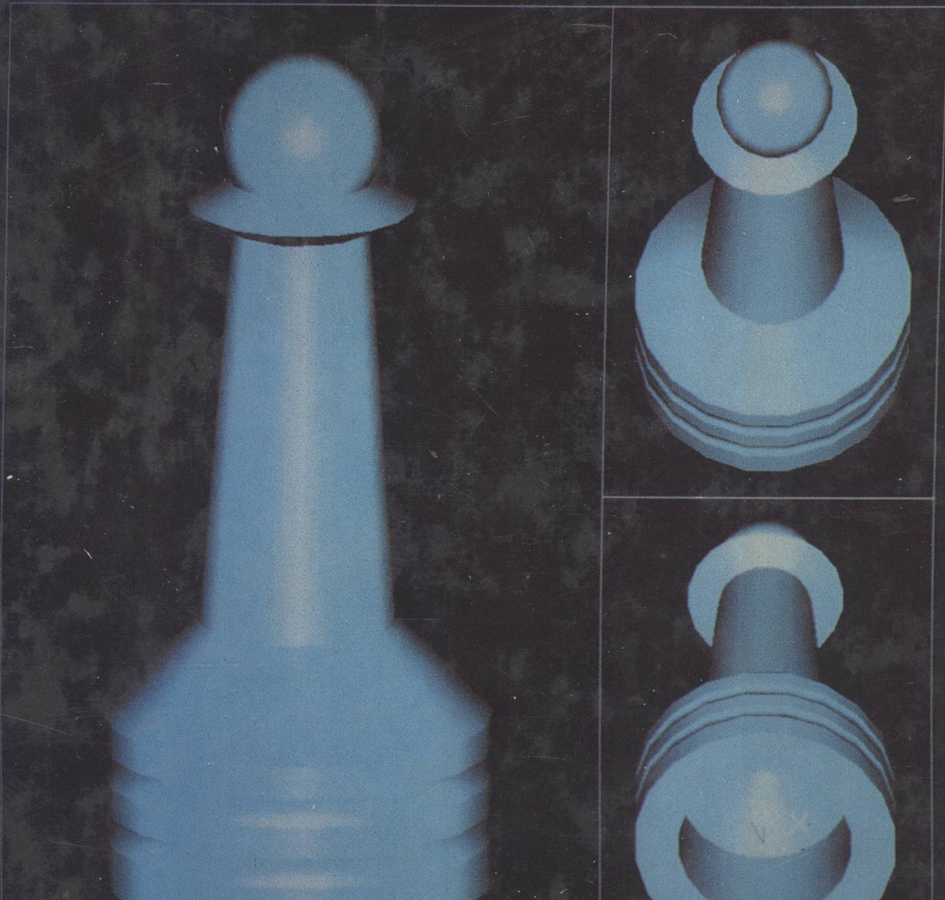
FINITE ELEMENT ANALYSIS ON MICROCOMPUTERS



NICHOLAS M. BARAN

Finite Element Analysis on Microcomputers

Nicholas M. Baran

McGraw-Hill Book Company

New York St. Louis San Francisco Auckland Bogotá
Hamburg London Madrid Mexico Milan
Montreal New Delhi Panama Paris São Paulo
Singapore Sydney Tokyo Toronto

Library of Congress Cataloging-in-Publication Data

Baran, Nicholas.
Finite element analysis on microcomputers.

Includes index.
1. Finite element method—Data processing.
2. Microcomputers—Programming. I. Title.
TA347.F5B355 1987 620′.001′515353 87-3251
ISBN 0-07-033694-6

1234567890 DOC/DOC 893210987

ISBN 0-07-033694-6

The editors for this book were Betty Sun and Marci Nugent, the designer was Naomi Auerbach, and the production supervisor was Richard Ausburn. It was set in Century Schoolbook by the McGraw-Hill Book Company Professional & Reference Division composition unit.

Printed and bound by R. R. Donnelley & Sons Company.

Finite Element Analysis on Microcomputers

In memory of my father, Paul Alexander Baran, 1910–1964

Preface

This book is an introductory text on finite element analysis with an emphasis on microcomputers. The intent of the book is to provide a practical text geared toward finite element users rather than toward students of finite element theory.

Examples in the text were developed using three microcomputer finite element programs: ANSYS-PC/LINEAR, Revision 4.2-A2, and ANSYS-PC/THERMAL, Revision 4.2.-0, both from Swanson Analysis Systems, Inc., and MSC/pal 2, Version 1.0, from The MacNeal-Schwendler Corporation. More recent versions of these software packages may now be available which may have different performance characteristics. The book's currency should not be affected, however, since the emphasis is on finite element concepts rather than specific software applications.

It should be noted that most of the computer-generated graphics from MSC/pal 2 were generated with the high-resolution Japanese IBM 5550 Kanji Model Computer and the IBM 5577 printer, thanks to Ken Blakely of The MacNeal-Schwendler Corporation. The graphics from ANSYS-PC/LINEAR and ANSYS-PC/THERMAL were generated with the much lower resolution IBM PC color graphics adapter and the IBM ProPrinter. The difference in quality of these graphics is not intended in any way to reflect on the actual products represented. ANSYS-PC/LINEAR and ANSYS-PC/THERMAL also support high-resolution graphics, but no equipment of this kind was available for use with these products.

Many generous people contributed to this project. Above all, I would like to thank Ken Blakely of The MacNeal-Schwendler Corporation and Laurie Scott of Swanson Analysis Systems, Inc. Without their generous help and cooperation, this book would not have been possible. I would also like to thank Dave Dietrich and Robert Gorman of Swanson Analysis Systems, Inc., and Mark Baker for reviewing parts of the manuscript. In addition, thanks are due Mike Bussler of Algor Interactive Systems, Jim Leung of Celestial Software, Richard Ay of COADE/McGraw-Hill, and Susan Tellep of Control Data. I am also indebted to Jan Benes and Tomas Frank, who both had a great deal to do with the publication of this book. Finally, I would like to thank my sponsoring editor, Betty Sun, for her unswerving encouragement throughout. And, to my wife Esther, thanks for the support.

Nicholas M. Baran

Trademark List

ANSYS	Swanson Analysis Systems, Inc.
ANSYS-PC/LINEAR	Swanson Analysis Systems, Inc.
ANSYS-PC/THERMAL	Swanson Analysis Systems, Inc.
MSC/pal 2	The MacNeal-Schwendler Corporation
MSC/AutoFEM	The MacNeal-Schwendler Corporation
NASTRAN	The MacNeal-Schwendler Corporation
IMAGES 3D	Celestial Software, Inc.
SUPERSAP	Algor Interactive Systems, Inc.
SAP-86	Number Crunching Systems, Inc.
SAP 80/81	Computers and Structures, Inc.
VAX 11-780	Digital Equipment Corp.
IBM PC	International Business Machines, Inc.
Apple Macintosh	Apple Computer, Inc.
FASTBACK	Fifth Generation Systems, Inc.
Apollo Domain	Apollo Computer, Inc.
Sun-3	Sun Microsystems, Inc.
Cyber 180	Control Data, Inc.
FINITE/GP	COADE/McGraw-Hill, Inc.
MARC	MARC Analysis, Inc.
GT-STRUDL	Georgia Institute of Technology
STARDYNE	System Development Corp.
IRMA Board	Digital Communications Associates
CROSSTALK	Microstuf, Inc.
Microsoft Mouse	Microsoft, Inc.
AutoCad	AutoDesk, Inc.

Introduction

This is a book for practicing engineers using the finite element method in their daily work. In addition to the basic concepts of the finite element method, the book provides the engineer with practical techniques and guidelines for obtaining accurate and useful results from finite element analysis. Microcomputer-based finite element programs are becoming increasingly popular in the industry, and, therefore, the emphasis in this book is placed on the use of the microcomputer for analysis work. However, the concepts and techniques presented are equally applicable to mainframe- and minicomputer-based programs.

The finite element method is probably the most widely used form of computer-based engineering analysis. Most engineers, from all disciplines, come across the finite element method at some point in their careers. The method is used for analyzing a broad range of engineering structures and components, from the human body to the wings of an airplane.

With the advent of the microcomputer, more and more engineers are gaining access to finite element analysis programs and are using them to solve an increasingly broad range of problems. While the finite element method is extremely useful, like any other tool, it can be misused, leading to inaccurate or inefficient results. Unfortunately, there are few, if any, books on the market that provide practical guidelines for *users* of the finite element method. There are a number of excellent books on the theory of the finite element method which generally provide a highly mathematical and rigorous treatment of the subject and are useful for those who plan to specialize in finite element analysis or intend to write their own finite element programs.

However, the great majority of engineers who use the finite element method are neither finite element specialists nor programmers interested in designing their own finite element programs. Most users employ finite element analysis as a tool for analyzing the adequacy of a structure or component under a variety of loading conditions. Many users use finite element analysis fairly infrequently and need a practical reference guide to refresh their memories on the important concepts and methodologies of finite element analysis. Others have just started working in industry and have never had a finite element class in college. This book is intended to fill the need for a practical guide that provides the methods and techniques necessary for effective and accurate use of finite element analysis. Some of the basic questions answered by this book are listed below:

1. What is *finite element analysis,* and how is it used?
2. What finite element codes are available, and where do microcomputers fit in?
3. What factors do I need to be aware of when I design my finite element model (e.g., element types, loading conditions, material properties, geometry, and boundary conditions?)
4. Is my problem suitable for running on a microcomputer? Can I download and/or upload programs from the mainframe?
5. What can go wrong with my analysis? How do I correct inaccurate results?
6. How do I interpret the results of my analysis?

Chapter 1 starts with a basic introduction to the finite element method and includes a series of examples of typical problems. Chapter 2 covers the types of structures and components that can be analyzed using the finite element method and how the various types of elements can be applied. The various types of loading conditions are also discussed. In Chap. 3, we take a look at the differences between mainframe, mini, and microcomputers and survey the major finite element codes in use today.

Starting with Chap. 4, we begin actually working through some finite element problems. We discuss basic modeling techniques: what to watch out for when designing a finite element model, what to avoid, and how to increase efficiency and accuracy. Chapters 5 and 6 work step by step through a finite element analysis. Chapters 7, 8, and 9 present a series of examples from the main areas of finite element analysis: statics, dynamics, and thermal (heat transfer) analyses.

It is assumed that the reader has a solid background in strength of materials and engineering mechanics. No knowledge of the finite element method is required. In order to use microcomputer-based finite element programs effectively, it is necessary to be familiar with the computer's operating system and with a simple text editor. These topics are discussed further in Chap. 3 and in App. A.

Contents

Chapter

1

Overview of Finite Element Analysis

Finite element methods are predominantly used to perform computer-based analyses of the static, dynamic, or thermal behavior of physical systems, structures, and components. They are used primarily when hand calculations cannot provide sufficiently accurate or detailed results or when the system to be analyzed is so complex that hand calculations are not appropriate. With the reduced cost of processing on microcomputers and "friendlier" software, finite element analysis has also become a viable alternative for solving small engineering problems that were previously solved "by hand."

Finite element analysis is broadly defined as a group of numerical methods for approximating the governing equations of any continuous system. In this book, we will consider its application to solids.

This chapter presents a basic introduction to the concepts and terminology of finite element analysis. Before discussing the theory of finite elements, it is important to emphasize that a thorough theoretical understanding of the finite element method is not a prerequisite to using finite element programs. The principle of finite elements, as one would expect, is a rigorous mathematical theory based on the calculus of variations, energy theorems, the principle of elasticity, and other equations of physics and engineering. It is not the purpose of this book to derive the theory of finite elements. There are many excellent texts available that cover the theory thoroughly, some of which are listed in the references at the end of this chapter.

Mastery of the finite element theory is absolutely essential if you are writing your own finite element programs. As a user of finite element programs, however, it is sufficient to have a rudimentary understanding of the basic concepts of the finite element method, which we will try to impart in this chapter. On the other hand, it is imperative as a user to understand the physical problem you are attempting to solve and how to use the finite element program and interpret its results correctly. The ultimate goal of this book is to impart this understanding.

In this chapter, we will first discuss the basic theory of finite element analysis. We will then work through the solution of a simple structural problem using the finite element method, both by hand and using a finite element computer program. This exercise will help clarify how finite element programs work and how they are used.

1.1 The Basic Concept of the Finite Element Method

The theory of finite elements is sometimes called the "theory of piecewise continuous approximations." In general, the objective of finite element analysis is to approximate with a sufficient degree of accuracy the values of the unknowns of a governing differential equation at selected points on the domain of a continuous physical system or structure. A mathematical model of the physical system or structure, divided into *nodes* and *finite elements*, is created, and the governing equations are applied to it and solved for each node.

The governing differential equation can define a wide variety of physical phenomena. Poisson's equation, for example, is a second-order partial differential equation which governs deflections of a membrane, bending of a prismatic beam, heat conduction with sources, and many other physical phenomena. The main function of the finite element program is to reduce the differential equation to a set of simultaneous algebraic equations which can be readily solved by a computer. The solution of these equations yields directly, or by means of minor additional computation, the desired unknown quantities, such as deflections, temperatures, or stresses.

The steps involved in a finite element analysis are shown in Table 1.1. The first step is to create the finite element model. The finite element model is a geometrical representation of the actual physical structure or body being analyzed. The model is created by dividing the structure into a number of subregions called "elements." The values of the unknown quantities are to be computed at selected points in the elements, usually at the corners. These points are called "nodes." Figure 1.1 shows an arbitrary shape divided into nodes and elements. The process of dividing up the body is often called "discretization" and is normally performed by the user. As we shall see in later chapters, discretization of the body or structure is the most important phase of the analysis and greatly affects the accuracy of the results.

TABLE 1.1. The steps in a finite element analysis

1. User creates the finite element model
   a. Define geometry, nodes, and elements
   b. Specify material properties, loading conditions, and boundary conditions
2. Finite element program performs analysis
   a. Formulate equation
   b. Solve equation
3. Finite element program reports results
   a. Compute node and element values (displacements, temperatures, stresses, reaction forces, etc.)
   b. Postprocess results (plots, code checks, etc.)

In addition to defining the location of nodes and elements, the user usually supplies the geometrical properties of the elements, material properties, boundary conditions, and loading conditions relevant to the analysis. Some finite element programs include or allow the use of databases that automatically supply the properties of standard structural elements.

It is important to emphasize that the finite element model is a mathematical simulation of the actual physical structure or body that it represents. The physical properties must be specified. If the structure is composed of I beams, for example, the elements must be assigned the geometrical properties of the I beam. If the body is made of steel, the material properties of

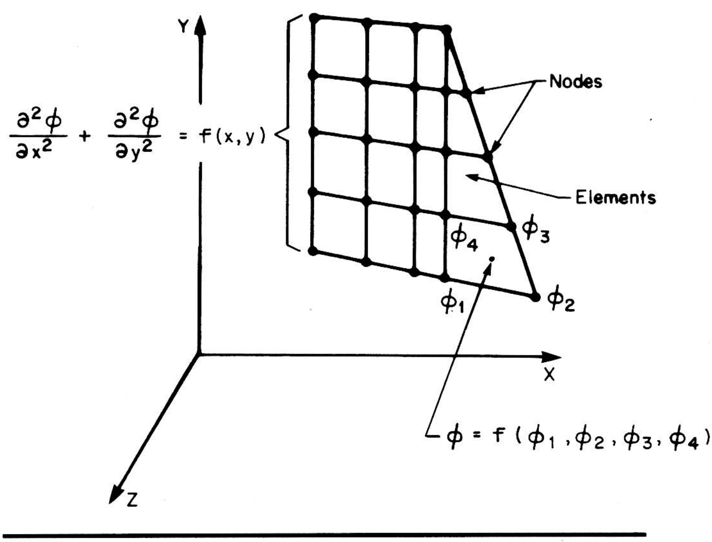


Figure 1.1 An arbitrary shape divided into nodes and elements. The shape is governed by the partial differential equation shown. The value of this equation at any point in an element is a function of the values of the nodes ϕ_i bounding the element.

steel must be assigned to the elements of the body. If the body is pinned at a support point or rigidly fixed, this support condition must be represented in the finite element model. Finally, the applied loads must be modeled. For example, a seismic analysis might require a lateral acceleration of 3 G to be applied to the model. A heat transfer analysis might require a temperature change in the working fluid of 300° over a period of 10 seconds. A static analysis may simply include the dead load of the structure.

In the second step of the analysis, the finite element program applies the governing partial differential equation and its boundary conditions in the form of an *equivalent integral formulation.* This is a procedure involving minimization of a functional (variational calculus) and energy conservation laws. For example, most finite element programs for structural analysis use the principle of virtual displacements to express the differential equations of equilibrium in their equivalent integral formulation. Further discussion of this procedure can be found in Bathe and Wilson, *Numerical Methods in Finite Element Analysis* (Ref. 1). Finite element programs are designed to work with specific differential equations. A program designed to solve heat transfer problems cannot solve structural analysis problems. However, many programs include several "modules," each designed to solve a certain type of problem.

After the finite element model is completely defined, the main numerical, or "number-crunching," phase of the analysis is performed by the finite element program. The program treats the nodal displacements as variables of an interpolation function, usually a polynomial, to give an analytical expression for displacement at any point inside the element. This is conceptually illustrated in Fig. 1.1. A polynomial (sometimes differential) function must be formulated for each element in the body. The polynomials are then substituted into the integral formulation of the partial differential equation resulting in a set of simultaneous algebraic equations which are solved to give the nodal values of the unknowns.

This may be the final step in the analysis or may be followed by additional computation where the nodal values are used to calculate other quantities. For example, in most finite element programs for structural analysis, the calculated nodal values are the displacements of the body. These displacements can then be used to calculate the strains and stresses in each element.

So far, we have discussed the concept of finite element analysis in very general terms. As we mentioned earlier, implementation of the finite element concept varies depending on the type of problem that is to be solved. Most finite element programs for microcomputers are designed to solve statics and dynamics problems in the elastic range. The primary reason for this is that these types of problems are the most straightforward to solve with finite element analysis and therefore require less