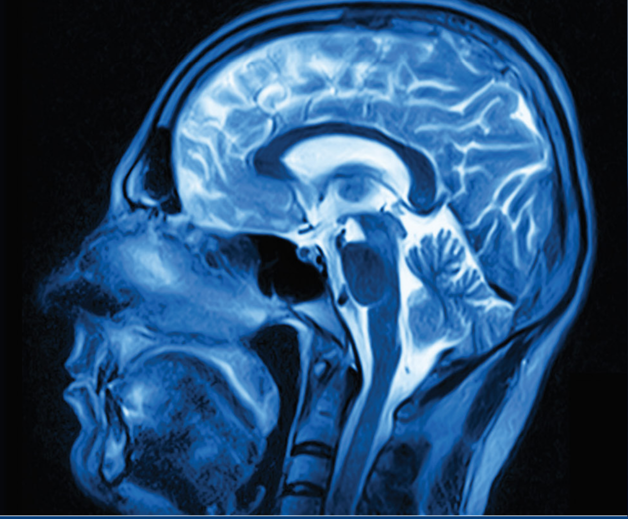
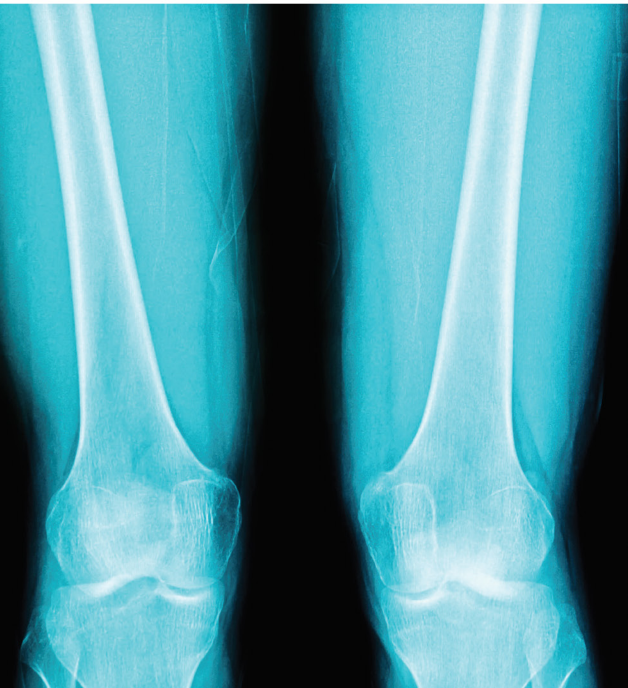
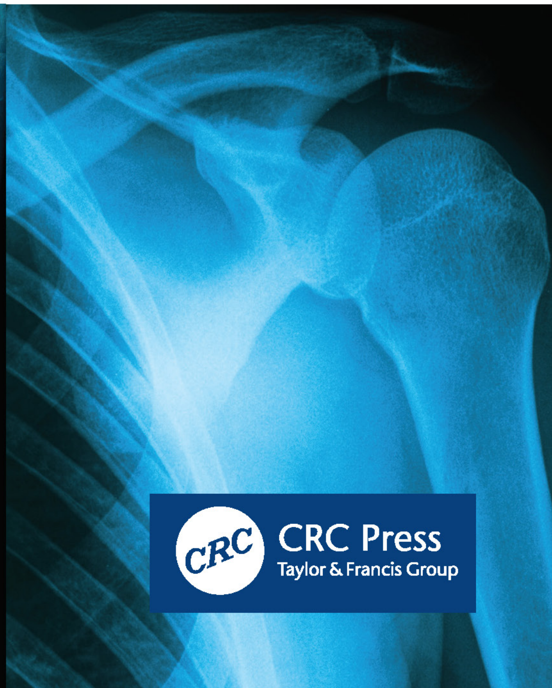


FINITE ELEMENT ANALYSIS FOR BIOMEDICAL ENGINEERING APPLICATIONS

Z. Yang





CRC Press
Taylor & Francis Group

Finite Element Analysis for Biomedical Engineering Applications

Finite Element Analysis for Biomedical Engineering Applications

Z. Yang

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

Printed on acid-free paper

International Standard Book Number-13: 978-0-367-18218-2 (Hardback)

Visit the Taylor & Francis Web site at
http://www.taylorandfrancis.com

and the CRC Press Web site at
http://www.crcpress.com

Contents

PART IV Simulation of Implants

PART V Retrospective

Preface

In 2001, I came to the University of Pittsburgh to pursue my PhD. As I learned about biomechanics, I became fascinated by the complications of biology. In the past 17 years, I had been working on many bioengineering projects with professors from the University of Pittsburgh, University of Pennsylvania, Allegheny General Hospital, and Soochow University. My long-time research has given me experience in finite element modeling in the field of biomedical studies. I have chosen to record my experiences in a book which, I hope, will encourage medical researchers to do further investigations. Yet, even after 17 years of study and research, I recognize that I still have more to learn about biomechanics. Should this book, therefore, contain errors, I ask readers to point them out to me so that I can address and correct them.

While I wrote this book, I received help and encouragement from many of my friends, including Frank Marx, Dr. J.S. Lin, Dr. Richard Debski, and Fayan Xu. Dr. Zhi-Hong Mao reviewed the whole manuscript. I am grateful for his constructive comments that have greatly improved the quality of the book. I give a special thanks to Ronna Edelstein for her time and effort in revising my manuscript. I express my great appreciation to the staff at CRC Press, especially Marc Gutierrez and Kari Budyk for their assistance in publishing the book. Finally, I thank my family, especially my wife, Peng, and my two children, for their constant support.

About the Author

Z. Yang earned a PhD in mechanical engineering at the University of Pittsburgh in 2004. Over the last 17 years, he has collaborated with professors from various colleges, such as the University of Pennsylvania and University of Pittsburgh, and finished a number of biomedical projects. Currently, he is a senior software engineer in the field of finite element analysis with over 10 years' experience.

1 Introduction

Because people are living longer in today's world, more individuals are dealing with a variety of diseases. Some common diseases are associated with the mechanical states of human organs. For example, hips often break when older people fall, and the lumbar disc degenerates due to excessive loadings over the long term. An abdominal aortic aneurysm (AAA) occurs when the stresses of the AAA wall exceed the strength of the wall tissue. Treatment of these diseases requires an understanding of the stress-states of relevant parts under various conditions. When some parts of the human body degenerate and lose their function, people may have to undergo implant surgeries, such as stent implantation for treatment of atherosclerosis and total knee replacement to regain the walking function. Although these implants can improve the person's quality of life significantly, they can also raise other issues, such as medial tilting in ankle replacements and fatigue and wear of the liner in hip implants. To solve these issues and improve the medical designs, it is vital to study the mechanical behavior of the implants.

While researchers are testing the mechanical responses of the organs and the implants in the lab, they also emphasize numerical simulations, especially finite element analysis. Since the 1970s, some well-known commercial finite element codes, such as ANSYS, NASTRAN, MARC, ABAQUS, LSDYNA, and COMSOL, have been developed to solve the structural problems. Among them, ANSYS software has the most powerful nonlinear solver, and hence it has become the most widely used software in both academia and industry. Over the past decade, many advanced finite element technologies have been developed in ANSYS. The purpose of this book is to simulate some common medical problems using finite element advanced technologies, which paves a path for medical researchers to perform further studies.

The book consists of four main parts. Each part begins by presenting the structure and function of the biology, and then it introduces the corresponding ANSYS advanced features. The final discussion highlights some specific biomedical problems simulated by ANSYS advanced features.

The topic of Part I is bone. After this introductory chapter, Chapter 2 introduces the structure and material properties of bone. Chapter 3 discusses the nonhomogeneous character of bone, including modeling it by computed tomography (CT) in Section 3.1 and by multidimensional interpolation in Section 3.2. Chapter 4 describes how to build a finite element model of anisotropic bone, and the crack-growth in the microstructure of cortical bone is simulated by eXtended Finite Element Model (XFEM) in Chapter 5.

Part II, which deals with soft tissues, is very detailed. Chapter 6 introduces the structure and material properties of soft tissues like cartilage, ligament, and intervertebral discs (IVDs). Next, Chapter 7 presents the nonlinear behavior of soft tissues and simulation of AAA in ANSYS190. Chapter 8 examines the viscoelasticity of soft tissues, including its application to the study of periodontal ligament creep.

Some soft tissues are enhanced by fibers. Chapter 9 discusses three approaches of fiber enhancement in ANSYS190: (1) standard mesh-dependent fiber enhancement, in which the fibers are created within the regular base mesh; (2) mesh-independent fiber enhancement that creates fibers independent of the base mesh; and (3) the anisotropic material model with fiber enhancement. The first two approaches are utilized to simulate the fibers in the annulus of the intervertebral disc (IVD).

Many nonlinear material models in ANSYS are available for the simulation of soft tissues. If the experimental data of one biological material do not fit any of these models, the researchers may turn to USERMAT in ANSYS. Chapter 10 focuses on the topic of how to develop user material models in ANSYS.

The soft tissues are biphasic, consisting of 30%–70% water. Chapter 11 introduces ways of modeling soft tissues as porous media and the application of biphasic modeling in head impact and IVD creep research.

Part III describes joint simulation. After briefly introducing the structure of joints in Chapter 12, in the next chapter, Section 13.1 defines three contact types in a whole-knee simulation, and a two-dimensional (2D) axisymmetrical poroelastic knee model is built in Section 13.2. Then, the discrete element method of knee joint that is implemented in ANSYS is analyzed in Chapter 14.

Part IV presents a number of implant simulations. Chapter 15 studies the contact of the talar component and the bone to investigate medial tilting in ankle replacement. The stent implantation is simulated in Chapter 16 using the shape memory alloy superelasticity model. The Archard wear model is applied to study the wear of the hip implant in Chapter 17. Chapter 18 predicts the fatigue life of a mini-dental implant using ANSYS SMART technology.

Chapter 19 presents a retrospective look at the entire content of the book. Some guidelines are summarized for the simulation of biomedical problems.

The biomedical problems in this book have been simulated using ANSYS Parametric Design Language (APDL). Reading this book requires knowledge of APDL. To learn APDL, I suggest first reading the ANSYS help documentation and then practice some technical demonstration problems available in this documentation. All APDL input files of the finite element models in the book are provided in the appendixes.