INFORMATION TO USERS

This manuscript has been reproduced from the microfilm master. UMI films the text directly from the original or copy submitted. Thus, some thesis and dissertation copies are in typewriter face, while others may be from any type of computer printer.

The quality of this reproduction is dependent upon the quality of the copy submitted. Broken or indistinct print, colored or poor quality illustrations and photographs, print bleedthrough, substandard margins, and improper alignment can adversely affect reproduction.

In the unlikely event that the author did not send UMI a complete manuscript and there are missing pages, these will be noted. Also, if unauthorized copyright material had to be removed, a note will indicate the deletion.

Oversize materials (e.g., maps, drawings, charts) are reproduced by sectioning the original, beginning at the upper left-hand corner and continuing from left to right in equal sections with small overlaps. Each original is also photographed in one exposure and is included in reduced form at the back of the book.

Photographs included in the original manuscript have been reproduced xerographically in this copy. Higher quality 6" x 9" black and white photographic prints are available for any photographs or illustrations appearing in this copy for an additional charge. Contact UMI directly to order.

UMI
University Microfilms International
A Bell & Howell Information Company
300 North Zeeb Road, Ann Arbor, MI 48106-1346 USA
313/761-4700  800/521-0600
Linear and non-linear finite element modeling of bone-implant system in uncemented total hip arthroplasty

Hou, Fu Joseph, Ph.D.
Rice University, 1993
RICE UNIVERSITY

Linear and Non-linear Finite Element Modeling of Bone-Implant System in Uncemented Total Hip Arthroplasty

by

Fu Joseph Hou

A Thesis Submitted in Partial Fulfillment of the Requirements for the Degree Doctor of Philosophy

Approved, Thesis Committee:

[Signatures]

Dr. J. Ed Akin, Director
Professor, Mechanical Engineering and Materials Science

Dr. Jeffrey D. Reuben
Adjunct Associate Professor, Mechanical Engineering and Materials Science

Dr. Angelo Miele
Professor, Mechanical Engineering and Materials Science

Dr. Panos C. Dakoulas
Associate Professor, Civil Engineering

Houston, Texas
April, 1993
Linear and Non-linear Finite Element Modeling of Bone-Implant System in Uncemented Total Hip Arthroplasty

Fu Joseph Hou

Abstract

Factors involved in prosthesis performance in uncemented total hip arthroplasty (THA) are complex and many of their effects unclear. Linear and non-linear models were developed to study effects of different factors involved in the bone-implant system.

A three-dimensional (3D) unsymmetric beam model, based on a potential energy formulation, for the analysis of stresses in long bones was developed in this research. The cross-sectional properties of the bone were obtained directly from computer tomography (CT) scan data. Variable cross-sectional properties were used along the beam axis. The formulation of the torsional element was based on the torsional theory of thin-walled closed sections and the flexural element bending effects were coupled in two orthogonal directions of unsymmetric bending. Several examples demonstrate that the beam model is efficient and valid in predicting stress profiles in long bones with or without a prosthesis.

A systematic method was developed to construct detailed, 3D solid finite element (FE) models from CT scan data. Strains from an intact femur model constructed by this method showed good agreement when compared to published experimental data. The model was also checked by an error estimator for its accuracy. Results from an idealized symmetric 3D model, including non-linear interface elements to
model the friction behavior of the bone-implant interface, showed that fully bonded and frictionless assumptions for the interface condition make quite a difference when compared to the friction case. To more accurately model the bone-implant system and to address the problem of implant stability, friction behavior in the interface should be included in the FE model. Through a study of an anatomically realistic bone-implant model under single-leg stance and stair climbing loads, it was shown that stair climbing loads are more threatening than single-leg stance loads to the prosthesis stability, especially the torsional stability. Stair climbing loads should be applied to all uncemented prosthetic devices before their implantations to test their stability in bone. Torsional stability should be rigorously pursued in addition to axial stability for uncemented prosthetic devices.
Acknowledgments

I would like to express my sincere appreciation to Dr. Ed Akin for his guidance and advice throughout this research. I also want to thank Dr. Jeffrey Reuben for providing advice from a medical perspective of view and partial financial aid. I am grateful to Dr. Angelo Miele and Dr. Panos Dakoulas for serving on the thesis committee.

I treasure the friendship with Dr. and Mrs. Shiou-Jyh Hwu and Dr. Harvey Phillips and the moments we spent together the past few years. Their encouragement and prayer were always there whenever I needed them. Encouragement and concern from friends, especially those from Mandarin Speaking Student Fellowship are also acknowledged.

Encouragement, support, and love from my parents and brothers were always driving forces for me to go on. I am also thankful for the support from my in-laws by whom I feel so blessed. Most of all, I want to thank my wife Hsin-Fang for accompanying me went through the years with understanding and unfailing love. Without her, all these would have been impossible.
Table of Contents

Abstract ii
Acknowledgments iv
List of Illustrations viii

1 Introduction 1
1.1 Motivation ................................................. 1
1.2 Contemporary Uncemented Total Hip Arthroplasty ...... 2
1.3 Finite Element in Total Hip Arthroplasty ............... 4
1.4 Objectives ................................................ 5
1.5 Organization ............................................. 6

2 A Three-Dimensional Unsymmetric Beam Model 8
2.1 Introduction ............................................... 8
2.2 Literature Survey ........................................... 9
2.3 Formulation of Element Stiffness Matrix ............... 10
2.3.1 Axial Element .......................................... 12
2.3.2 Thin-Walled Torsional Element ...................... 14
2.3.3 Flexural Element ...................................... 17
2.3.4 Assembly of Element Stiffness Matrix $K^e$ .... 22
2.3.5 Transformation of Element Stiffness Matrix .... 22
2.3.6 Reaction and Stress Recovery ...................... 22
2.4 Transformation of Coordinates ......................... 24
2.5 Model Verification ...................................... 27
2.5.1 Bending of a Rectangular Cantilever Beam .......... 27
2.5.2 Torsion of a Tapered Thin-Walled Cantilever Beam .... 29
2.5.3 Unsymmetrical Bending of a Z-shaped Cantilever Beam .. 29
2.5.4 Bending of a Tapered, Variable Cross-sectioned Cantilever Beam 33
2.5.5 Bending and Torsion of a Composite Beam ............ 33
2.6 Summary ............................................. 38

3 Three-Dimensional Stress Analysis of an Intact Femur 39

3.1 Introduction ......................................... 39
3.2 Literature Review .................................... 40
3.3 Methods ............................................. 42
  3.3.1 Geometric Model ................................ 42
  3.3.2 Finite Element Modeling ....................... 45
3.4 Model Verification .................................. 50
  3.4.1 Comparison with Experimental Results .......... 50
  3.4.2 Error Estimate .................................. 57
3.5 Comparison with Beam Model Results .................. 58
  3.5.1 Load Calculation During Single-Leg Stance ... 58
  3.5.2 Beam Model of an Intact Femur ............... 60
  3.5.3 Results and Discussion ......................... 61
3.6 Summary ............................................. 63

4 Non-linear Interface Analysis of an Idealized Symmetric
Bone-Implant Model .................................. 66

4.1 Introduction ......................................... 66
4.2 Literature Review .................................... 67
4.3 Methods ............................................. 69
5 Implant Stability Study Using an Anatomically Realistic Non-linear Bone-Implant Model

5.1 Introduction .............................................. 83
5.2 Literature Review ......................................... 84
5.3 Methods ...................................................... 86
  5.3.1 Geometric Model ........................................ 86
  5.3.2 Finite Element Mesh .................................... 86
  5.3.3 Loads, Boundary Conditions, and Material Properties 87
5.4 Results ..................................................... 93
5.5 Discussions ................................................ 101
5.6 Comparison with Beam Model Results ..................... 104
5.7 Summary and Conclusion ................................. 109

6 Conclusions .................................................. 110

6.1 Summary ................................................... 110
6.2 Concluding Remarks ...................................... 111

A Verification of the Non-linear Interface Element .............. 114

Bibliography .................................................... 120
List of Figures

2.1 Local axes and force components of the beam element. .............. 11
2.2 Displacement and force component of an axial element. ............ 13
2.3 A closed thin-walled cross-section. .................................. 15
2.4 Displacement and force components of a flexural element. .......... 18
2.5 Submatrices $K_{11}$, $K_{21}$, and $K_{22}$ of the stiffness matrix. ....... 23
2.6 Transformation between local axes and global axes. .................. 25
2.7 Dimension of a rectangular cantilever beam .......................... 28
2.8 Bending moment and bending stress in a rectangular cantilever beam. 30
2.9 Dimension and angle of twist of a thin-walled cantilever beam. ... 31
2.10 Cross-section of a Z-shaped cantilever beam. .......................... 32
2.11 Dimension and load of a variable cross-sectioned cantilever beam. .. 34
2.12 Comparison of bending stresses from beam model and ACSL. ....... 35
2.13 Dimension and loads of a composite cantilever beam. ............... 36
2.14 Comparison of displacement and bending stresses in the composite
    beam. ........................................................................... 37

3.1 Flow chart of the general procedure to construct the 3D solid model. 43
3.2 Anatomic directions of a right intact femur. ............................. 44
3.3 Parametric lines and hidden line plot of the geometric model of the
    intact femur. ................................................................... 46
3.4 Element groups for (a) solid cortical bone (b) solid trabecular bone and (c) thin shell cortical bone. ........................................ 48
3.5 Loading conditions for (a) inclined axial load and (b) torsional load. ......................................................... 51
3.6 Strain gage levels on the intact femur model ................................................................. 53
3.7 Comparison of strain values between FE model and strain gage under inclined axial load. ........................................ 54
3.8 Comparison of strain values between FE model and strain gage under torsional load. ................................................ 55
3.9 Comparison of strain values between FE model and strain gage under combined torsional load and vertical load. .................. 56
3.10 Error estimate of the intact femur model under single-leg stance load. ....................................................... 59
3.11 Free body diagram and force equilibrium triangle under single-leg stance loads. .............................................. 61
3.12 Beam elements and cross-sectional contours for the intact femur model. ...................... 62
3.13 Comparison of von Mises stresses on medial and lateral cortex of the intact femur. ................. 64

4.1 Geometric primitives used in constructing the idealized symmetric bone-implant model. ......................... 70
4.2 Topology of the interface element INTER9. ................................................. 71
4.3 Element groups for the idealized bone-implant model. ................................................ 73
4.4 Regions of contact for frictionless case and friction case. ........................................... 75
4.5 Locations of sample nodes for relative motion calculation on the medial and lateral sides of the interface. ..................... 77
4.6 Comparison of relative motions for the frictionless case (case 1, \( \mu = 0. \)) and friction case (case 2, \( \mu = 0.47. \)) .................. 78
4.7 Normal stresses on medial and lateral sides of the implant. 80
4.8 Von Mises stresses in trabecular bone for the three interface conditions. 81

5.1 Parametric lines constructed from selected grids. 88
5.2 Hyperpatches for the geometric model. 89
5.3 Element groups for the FE model. 90
5.4 2D Free body diagram of femur during stair climbing. 92
5.5 Fringe plot of the estimated error on the bone-implant model with fully bonded interface under single-leg stance loads. 94
5.6 Regions of contact in the interface under two loading cases. 96
5.7 Locations of sample nodes for relative motion calculations. 97
5.8 Axial component of the relative motion under two loading cases. 98
5.9 Transverse component of the relative motion under two loading cases. 99
5.10 Total relative motion under two loading cases. 100
5.11 Von Mises stresses in the trabecular bone under single-leg stance loads. 102
5.12 Von Mises stresses in the trabecular bone under stair climbing loads. 103
5.13 Beam elements and cross-sectional contour lines for the anatomic bone-implant model. 106
5.14 Comparison of von Mises stresses between beam model and 3D solid model under single-leg stance loads. 107
5.15 Comparison of von Mises stresses between beam model and 3D solid model under stair climbing loads. 108

A.1 Dimension of the axisymmetric model. 115
A.2 Finite element mesh for the axisymmetric model. 116
A.3 Normal stresses in the stem. 118
A.4  Shear stresses in the stem. ................................. 119
Chapter 1

Introduction

1.1 Motivation

Joint replacement surgery restores movement and mobility, eliminates chronic pain, and helps patients return to most normal daily activities for thousands of people every year. According to the American Academy of Orthopedic Surgeons, there are between 250,000 and 300,000 joint replacements performed every year. Knee and hip replacements, respectively, account for 130,000 and 126,000 procedures annually. Shoulder replacements come in a distant third at 5,000.

Currently, there are only two widely accepted methods for securing hip replacements in place, polymethyl methacrylate cement (PMMA) used in cemented devices and press-fit for uncemented devices. The use of PMMA has its origin back to late 1950's. Although the successful rate with cemented devices is over 90% for short-term (under 10 years), it is now clear to most hip surgeons that the major long-term problem with hip replacement utilizing PMMA is loss of fixation, with the attendant problems of pain and loss of bone stock. This appears to be not only a mechanical problem but also a problem of the biologic reactivity of PMMA to bone. Younger patients, primarily those under 60 years of age, do not do well with cemented total hip arthroplasty (THA) due to the problems of overuse, weight, and compliance. Major efforts have been undertaken to develop fixation not dependent on PMMA.

With clinical trials it has been estimated that 2800 hips followed for 5 years or 700 followed for 10 years would be necessary to show a significant difference in clinical performance between cemented and uncemented stems [42]. It is likely that even more cases would be required to establish differences between similar uncemented
designs. This is an enormous imposition on the time and possible health of a large number of patients. It also places a large demand on the time and resources of the investigator. Furthermore, with today’s rapidly changing medical technology, there is a very likely chance that the long time spent on such clinical studies will only result in data on obsolete devices. This suggests the increasing need for a systematic and reliable method which can be done relatively quickly for parametric studies of various prosthesis design concepts. It is also very important that these studies should be done before any clinical implantation, so as to avoid subjecting patients to needless uncertainties.

1.2 Contemporary Uncemented Total Hip Arthroplasty

There are two major methods to secure the uncemented device in bone: press-fitted and porous-coated. The later one allows for bony ingrowth into the interstices of the porous coating thus providing biologic fixation.

Problems associated with uncemented prostheses arise from three sources. First, since no cement is used to fill the gaps between the prosthesis and the host bone, the dimensions and shape of the prosthesis in relation to the host bone become more important. However, the problem of producing a good fit has not been solved by the available contemporary prosthesis designs. Patients may suffer from postoperative pain (midthigh pain due to relative motions of the implant during loading [46]). These relative motions may also prevent bony ingrowth or osseous integration of the implant. This will affect the short-term and long-term stability of the device.

Second, strain-adaptive bone remodeling phenomena appears to be an important factor affecting the long-term behavior of uncemented THA. This is due to the stiff material used for the prosthesis. When a stiffer device is inserted in bone, most of the load which used to be carried by bone itself is now carried by the stiff prosthesis
instead. Certain regions of the surrounding bone would become understressed, thus causing bone resorption to take place. This phenomenon is called stress shielding effect. The long-term effect of this mechanism is not known. However, too much bone loss will eventually cause mechanical failure of the bone-implant structure. Third, prostheses that have been firmly fixed by bony ingrowth or by osseous integration are very difficult to remove if it becomes necessary. It is not certain that failed uncemented prostheses are more easily revised than failed cemented prostheses.

With these complicated problems, design and analysis of the hip prosthesis are evidently still in a development stage. More clinical observations combined with engineering analyses will further the knowledge and help in designing better devices. Assessment of stress distribution in the bone-implant system has been used to study component fracture, cement fracture, interface loosening, and stress related bone resorption and remodeling. These stresses depend on loading condition, geometry of the implant, material properties, and boundary (interface) conditions. Thus the stresses are influenced by implant design, choice of materials, and fixation methods.

The development of truly optimized hip implants is impeded by an imperfect understanding of many aspects of the complex behavior of the bone-implant structure. Stress analyses can help in adding an understanding of this complicated structure by quantifying and interrelating different parameters. It can be done analytically, experimentally, or combination of both. Finite element method can be used to predict objectively the mechanical performance of different devices on a relative basis.
1.3 Finite Element in Total Hip Arthroplasty

The finite element method (FEM) is firmly established as a powerful and popular analysis tool in engineering practice. It is a numerical procedure for analyzing structures and continua. The greatest advantage of using FEM in biomechanics is its power in doing parametric studies. One of the earliest applications of FEM to orthopedic biomechanics was introduced in 1972 by Brekelmans et al. [11]. During the past twenty years since its first introduction, FEM has become an indispensable tool for orthopedic researchers due to its capabilities in handling complex geometries, versatile material properties, complicated boundary conditions and load cases, and applicability to time dependent and non-linear problems. The method has been applied in orthopedic research to stress analysis of bone, artificial joint and fixation design, analysis of soft tissues, strain induced bone remodeling and thermal analysis of bone cement, etc. The numerous papers on these subjects show the interest and value of FEM to the researchers in this field.

Finite element (FE) models are approximations to a real system. Certain assumptions are made during the model building process according to the simulation purposes of the model. It is hardly seen in a model that contains every detail of the real situation. The complex anatomic geometry of the bone-implant structure can only be adequately accounted for by a three-dimensional (3D) solid model. Although a 3D solid model is preferred in most cases, the complexity and difficulties in building an accurate model and the computational time needed to solve the model limit its utility. A simple model is desirable for a quick parametric study of preliminary design concepts. A 3D FE beam model has advantages over 3D solid models in terms of simplicity, data preparation and computer costs.

In THA, failure of the procedure due to loosening of components continues to be acknowledged as the most important potential long-term complication. Further
understanding of the stress distribution and relative motion in the bone-implant interface are crucial to address the issue. Most FE models in the literatures assumed fully bonded or no-tension, no friction conditions for the interface. This is not the case especially in early post-operative period. It is desirable to know how these assumptions differ from the real situation in which friction is involved in the interface behavior. A FE model that includes friction effects may more accurately approximate the real condition.

In modeling the loading conditions, the torsional loads generated during activities such as stair climbing and rising from a seated position are suspected to be more threatening to the stability of the implant than axial loads generated during normal gait or single-leg stance. An understanding of the difference in performance of the implant under these two kinds of loading conditions will benefit the design of new devices with better stability.

1.4 Objectives

- Propose a simple model for quick parametric study of the bone-implant system.

- Develop a systematic method to construct detailed 3D FE models which accurately account for the complex geometry of the bone-implant structure and do detailed 3D stress analysis of the bone-implant system.

- Study the influences of different assumptions for the interface condition of the bone-implant interface on stress distribution and implant stability.

- Study the stability of one prosthetic device under loads simulating single-leg stance and stair climbing. Compare the debonding conditions under these two different loading cases.
1.5 Organization

An unsymmetric, 3D finite element beam model is developed in chapter 2 to serve as a preliminary design tool. The model uses computer tomography (CT) scan data for cross-sectional properties calculation. The beam model is verified for its accuracy in predicting stresses by several test examples with simple beam geometries.

In chapter 3, a method combining CT scan data and computer-aided design (CAD) techniques to build a detailed 3D FE solid model of an intact femur is developed. The purpose is to account for the complex geometry of the bone accurately. The accuracy of the model is studied by comparing its results with published experimental data and by checking its computational error with an error estimator. Stresses in the intact femur predicted by the proposed beam model are compared with results from the 3D solid model.

Non-linear interface analysis of an idealized symmetric bone-implant model is presented in chapter 4. Non-linear interface elements are included in the model to simulate various interface conditions. The model is used to study the general influences of different assumptions for bone-implant interface conditions on contact, relative motion, and stress distributions.

Stress analysis of an anatomically realistic bone-implant model under two different loading conditions is studied in chapter 5. The two loading cases simulate loads generated during single-leg stance and stair climbing. The purpose of this study is to assess the stability of the implant under these two kinds of loading condition. Three interface conditions are studied: fully bonded; no-tension, with friction simulating smooth prosthetic surface; and no-tension, with friction simulating porous-coated prosthetic surface. Stresses predicted by the beam model are compared with the 3D solid model with fully bonded bone-implant interface.
Chapter 6 summarizes the research. Appendix A contains verification of the non-linear interface element.
Chapter 2

A Three-Dimensional Unsymmetric Beam Model

2.1 Introduction

Biomechanical modeling of the human musculoskeletal system has been and remains one of the challenging tasks in bioengineering. The load bearing ability of bone, especially the long bone, is of great interest to biomechanical researchers. Stress analyses of long bones have been developed using beam theory, or beam on elastic foundation theory [7, 28, 37, 49, 51, 54, 56, 61], two-dimensional (2D) finite element method (FEM), and three-dimensional (3D) FEM [30]. In most cases, the bone is assumed to be an isotropic or transversely isotropic material, or a combination of two isotropic materials – cortical bone and trabecular bone.

The 3D finite element beam model approach has advantages over 2D and 3D finite element models in terms of simplicity and computer costs. Since it requires less computational effort, the stress analysis can be performed on a personal computer. The purpose of this chapter is to develop a 3D unsymmetric beam model for predicting stress profiles in the proximal femur before and after femoral head replacement. The model can serve as a preliminary tool for prosthetic design by easily analyzing a series of alternate design solutions. A 3D finite element solid model can then be used to determine the final optimal design. The model is constructed with a series of parallel cross-sections along the beam axis so that the cross-sectional properties can be obtained directly from CT scan data. Variable stiffness properties along the beam element are calculated directly from the CT scan data. The properties at the Gauss quadrature points are obtained by quadratic or spline interpolation. The stiffness terms for the element are calculated numerically by Gauss quadrature along
the length of the element. The torsional effect is approximated by the torsional theory of a thin-walled closed section. Unlike other beam models the unsymmetrical bending effects of the two orthogonal planes of bending are coupled together.

2.2 Literature Survey

Modeling of the long bone using beam theory is a natural approach. Previous works on the stress analysis of the long bone using beam theory can be traced back to early 1900's. The classical work of Koch [37] utilized basic beam theory to calculate the stresses induced in the femur subjected to various loading conditions. Toridis [61] analytically modeled the femur as a 3D space curve based on free body statics. Rybicki [54] analyzed the femur based on the beam model developed by Koch [37] and modeled the proximal third of the femur with a 2D, variable thickness finite element model. Piziali [49] used a 3D finite element beam model developed by Priezmienicki [50] to examine some approximation effects used by previous investigators. Huiskes [28] developed a model based on a beam on an elastic foundation theory to analyze stresses in a femoral-prosthetic system. Raftopoulos and Qassem [51] developed a mathematical model based on curved beam theory to analyze the stresses in the femur. The bone material was considered as both isotropic and anisotropic. Salathe [56] developed a generalized beam theory to analyze statically indeterminant problems involving long bones and applied it to the fifth metatarsal. For parametric evaluation of implant systems, Bechtold and Riley [7] developed a numerical technique for a beam on an elastic foundation model based on B-spline differential equation modeling.

Among the previous papers, only Toridis [61] mentioned using an unsymmetrical bending equation to recover bending stress. He used average structural properties along the length of the beam whereas the model described in this chapter uses variable properties along the beam axis. Since the material properties vary with the apparent
density of the bone and the density varies along the bone's longitudinal axis [14], using variable properties approximates the real situation more accurately. Torsional effects were considered by most authors except for the beam on an elastic foundation models [7, 28]. No analyses have used the torsional theory of a thin-walled closed section approach.

2.3 Formulation of Element Stiffness Matrix

The derivation of the beam element stiffness matrix will base on the potential energy formulation. The theoretical developments in structural and solid mechanics applications always produce a strain energy $U^e$ that has the matrix form

$$U^e = \frac{1}{2} u^e T K^e u^e$$  \hspace{1cm} (2.1)

where $u^e$ is the element displacements and $K^e$ is the symmetric element stiffness matrix.

The centroidal axis of a long bone is a curve in three-dimensions. This curve can be approximated by a series of straight beam elements. The 3D beam element is a combination of an axial element, a torsional element and a flexural element with unsymmetrical bending on two orthogonal planes. The beam element has two nodes on its two ends and each node has six degrees of freedom - three translations and three rotations. The local axes and force components of a beam element are illustrated in Figure 2.1. The $z_l$ axis is along the the long axis of the beam and $x_l$ and $y_l$ are in the transverse directions. The global axis system is chosen to be the same as that of the CT scan system. Therefore, it is natural to choose the $z$ axis in the long axis direction of bone. The formulations of the axial element, torsional element and flexural element are derived in the followings respectively.
Figure 2.1  Local axes and force components of the beam element.
2.3.1 Axial Element

An axial element is illustrated in Figure 2.2 with a single translation $u$ in the $z$ direction at any point. Assume that the displacement $u$ at any point within the element varies linearly with $z$. Let $L$ be the length of the element, the interpolation function $H$ can be expressed as

$$ H = \begin{bmatrix} H_1 & H_2 \end{bmatrix} = \begin{bmatrix} 1 - \frac{z}{L} & \frac{z}{L} \end{bmatrix}. $$

The element stiffness matrix for an axial element is

$$ K^e = \int_V B^T D B \, dv \quad (2.2) $$

where $B$ is the first derivative of $H$ with respect to $z$. Thus

$$ B = \frac{dH}{dz} = \frac{1}{L} \begin{bmatrix} -1 & 1 \end{bmatrix} $$

$D$ in this case is the Young’s modulus of the material. Rewriting Eqn. (2.2) gives

$$ K^e = \int_V B^T D B \, dv = \frac{1}{L^2} \begin{bmatrix} -1 & 1 \end{bmatrix} \int_L \int_A E \, da \, dz $$

Multiplication and integration over the cross-section gives

$$ K^e = \frac{1}{L^2} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \int_L \bar{E} A \, dz \quad (2.3) $$

where

$$ \bar{E} A = \int_A E \, da $$

is used since the bone’s $E$ varies significantly with location. In a given scan plane of $z = constant$, this integral is approximated as

$$ \bar{E} A = A_p \sum_{k=1}^{K} E_k $$
Figure 2.2 Displacement and force component of an axial element.

where $K$ is the number of pixel in the bone region and $A_p$ is the area of a typical pixel. The maximum value of $K$ is currently $512^2$. If there are $n$ quadrature planes along the length of an element, then

$$
K^e = \frac{1}{L^2_2} \sum_{q=1}^{n} \tilde{E}_q A_q \omega_q |J_q| \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix}
$$

where $\omega_q$ is the weight and $J_q$ is the Jacobian evaluated at quadrature coordinate $z_q$. The element spans two CT scan sections. The values of $\tilde{E}A$ are computed at the two ends of the element and at its midpoint. Then quadratic interpolation is used to define the values of $\tilde{E}A(z)$ at the Gauss quadrature points as

$$
\tilde{E}A(z_q) = \sum_{i=1}^{3} (\tilde{E}A)_i \Phi_i(z_q)
$$

where $\Phi$ is the quadratic Lagrangian interpolation function. The integral in Eqn. (2.3) is computed using three point Gaussian quadrature in the z-direction.
2.3.2 Thin-Walled Torsional Element

Since the Young's modulus of the cortical bone is two orders of magnitude larger than that of the trabecular bone, cortical bone contributes to the majority of the torsional rigidity of a cross-section. Therefore, the torsional behavior of a long bone cross-section can be approximated by the torsional theory of a thin-walled closed section. The torsional theory for thin-walled closed sections was first developed by Bredt [10] in 1896. The following derivation of the torsional element for long bone is mainly based on Bredt's theory and combined with the principle of strain energy.

For a closed thin-walled cross-section (Figure 2.3), the shear flow \( q = \tau t \) is constant everywhere along the perimeter. Here, \( \tau \) is the shear stress which is constant through the thickness and \( t \) is the thickness at a certain point on the perimeter where \( \tau \) is measured. It is assumed that the cross-section retains its shape during twisting so the warping effect is negligible. Consider the overall equilibrium between the applied twisting moment \( M_z \) and the moment of the internal shear stress. Referring to Figure 2.3, the force due to an element \( ds \) is \( qds \) and the moment it exerts about the center of twist is

\[
dM_z = hq \, ds \quad (2.4)
\]

where \( h \) is the perpendicular distance between \( ds \) and the center of twist. The area of the shaded area \( dA \) is

\[
dA = \frac{1}{2} h \, ds \quad . \quad (2.5)
\]

From Eqns. (2.4) and (2.5), we have

\[
dM_z = 2q \, dA \quad .
\]

Integration over the full cross section gives

\[
M_z = 2qA
\]
Figure 2.3 A closed thin-walled cross-section.
where $A$ is the total area enclosed by the median perimeter. The shear flow $q$ is then

$$q = \frac{M_z}{2A}.$$  \hfill (2.6)

The strain energy per unit volume due to the shear stress $\tau$ is $\tau^2/2G$ where $G$ is the shear modulus of the material. The strain energy per unit length due to the torsional shear in the cross-section with unit thickness is

$$dU = \int \frac{\tau^2}{2G} t \, ds.$$  \hfill (2.7)

Integration over the element length $L$ yields

$$U^e = \int_L \int \frac{\tau^2}{2G} t \, ds \, dz.$$  \hfill (2.8)

The external work, $W^e$, done by the torque $M_z$ over a differential length of the element $dz$ is equal to the strain energy generated in the element. Thus,

$$dW^e = \frac{1}{2} M_z \, d\theta = dz \int \frac{\tau^2}{2G} t \, ds.$$  \hfill (2.9)

Substituting Eqn. (2.9) into (2.8) yields

$$W^e = \frac{1}{2} \left[ \int L M_z \frac{d\theta}{dz} \, dz = \frac{1}{2} \int L M_z \theta' \, dz \right].$$  \hfill (2.10)

From Eqns. (2.7) and (2.9) and the definition of shear flow $q$ (Eqn. 2.6), we have

$$M_z \frac{d\theta}{dz} = \int \frac{\tau^2 t^2}{Gt} \, ds = q^2 \int \frac{ds}{Gt} = \frac{M_z^2}{4A^2} \int \frac{ds}{Gt}.$$  \hfill (2.11)

If $\tilde{G}$ is obtained by averaging over the cross-section, as

$$\frac{1}{G} \int \frac{ds}{t} \equiv \frac{1}{\tilde{G}} \int \frac{ds}{Gt}.$$

After rearranging, Eqn. (2.11) becomes

$$M_z = \tilde{G} \frac{4A^2}{\int \frac{ds}{t}} \theta' = \tilde{G} J \theta'.$$  \hfill (2.12)
where
\[ J = \frac{4A^2}{\int \frac{ds}{t}} \]  
(2.13)
is the St. Vernant's torsional constant for the cross-section. Substituting Eqn. (2.12) into Eqn. (2.10) yields
\[ U^e = \frac{1}{2} \int_L \bar{G} \theta^2 dz . \]  
(2.14)
Interpolating \( \theta \) by interpolation function \( H^e \) and letting \( B^e \) be the first derivative of \( H^e \) (\( H^{e'} = B^e \)), we have
\[ \theta = H^e \theta^e \quad \theta' = B^e \theta^e . \]  
(2.15)
Substituting Eqn. (2.15) into Eqn. (2.14) yields
\[ U^e = \frac{1}{2} \theta^e^T K^e_{\theta} \theta^e \]  
(2.16)
where
\[ K^e_{\theta} = \int_L B^e^T \bar{G} J B^e \ dz \]  
(2.17)
is the element stiffness matrix for the torsional element.

2.3.3 Flexural Element

For an unsymmetrical cross-section, the principal axes are in the direction where the product of inertia \( I_{xy} \) vanishes. For any other perpendicular axes on the cross-section, \( I_{xy} \) is not zero and has an effect on the bending stress. For a beam with unsymmetrical cross-sections, the bending effects are coupled in both transverse directions. The displacement and force components of a flexural element are shown in Figure 2.4.

Let \( u \) and \( v \) be the transverse displacement in \( x \) and \( y \) directions respectively. The axial displacement \( w \) in the \( z \) direction varies linearly with \( x \) and \( y \). The axial displacement relation, for small slopes, is
Figure 2.4 Displacement and force components of a flexural element.
\[ w(x, y, z) = xu' - yv' = x \frac{du}{dz} - y \frac{dv}{dz}. \]

The axial strain \( \varepsilon_z \) is
\[ \varepsilon_z = \frac{\partial w}{\partial z} = x \frac{d^2 u}{dz^2} - y \frac{d^2 v}{dz^2} = xu'' - yv''. \]

For an elastic material the stress, \( \sigma \), is defined by Hook's law as
\[ \sigma = E \varepsilon = E (xu'' - yv''). \]

The strain energy is defined as
\[ U = \frac{1}{2} \int_V \sigma \varepsilon \, dv = \frac{1}{2} \int_V E (xu'' - yv'') \left( xu'' - yv'' \right) \, dv. \]

Expanding and rearranging gives
\[ U = \frac{1}{2} \left( \int_V E x^2 u''^2 \, dv + \int_V E y^2 v''^2 \, dv \right) - \int_V E xy u'' v'' \, dv. \] (2.18)

Interpolating the displacements for a flexural element by interpolation function \( H^e \) gives
\[ u = H^e u^e \quad \text{and} \quad v = H^e v^e. \]

Let \( B^e \) be the second derivative of \( H^e \) (\( H^{e''} = B^e \)). Then the second derivatives of the displacement vectors \( u^e \) and \( v^e \) can be written as
\[ u'' = B^e u^e \quad \text{and} \quad v'' = B^e v^e. \] (2.19)

Substituting Eqn. (2.19) into Eqn. (2.18) gives
\[ U^e = \frac{1}{2} \left( u^{\varepsilon^T} K_1^e u^e + v^{\varepsilon^T} K_2^e v^e \right) - u^{\varepsilon^T} K_3^e v^e \]

where
\[
K_1^e = \int_L \int_A B^e T E x^2 B^e \, da \, dz = \int_L B^e T E I_y B^e \, dz \tag{2.20.a}
\]
\[
K_2^e = \int_L \int_A B^e T E y^2 B^e \, da \, dz = \int_L B^e T E I_x B^e \, dz \tag{2.20.b}
\]
\[
K_3^e = \int_L \int_A B^e T E x y B^e \, da \, dz = \int_L B^e T E I_{xy} B^e \, dz \tag{2.20.c}
\]

and
\[
\overline{EI}_y = \int_A E x^2 \, da
\]
\[
\overline{EI}_x = \int_A E y^2 \, da
\]
\[
\overline{EI}_{xy} = \int_A E x y \, da.
\]

These sectional properties are calculated at each given CT scan slice as described for \(\overline{E}A\). The properties at the Gauss quadrature points are obtained by quadratic interpolation in \(z\). \(U^e\) can also be expressed as
\[
U^e = \frac{1}{2} \left[ u^e T \left( K_1^e u^e - K_3^e v^e \right) + v^e T \left( K_2^e v^e - K_3^e u^e \right) \right].
\]

Or in matrix form
\[
U^e = \frac{1}{2} \begin{bmatrix} u^e & v^e \end{bmatrix} \begin{bmatrix} K_1^e & -K_3^e \\ -K_3^e & K_2^e \end{bmatrix} \begin{bmatrix} u^e \\ v^e \end{bmatrix}.
\tag{2.21}
\]

The element stiffness matrix of the flexural element is an \(8 \times 8\) matrix. The off-diagonal submatrices (i.e. \(-K_3^e\)) are the effects of the coupled bending of the two orthogonal planes.

The most common selection for the interpolation function for the flexural element is the cubic Hermite polynomials so that the deflection and the slope are continuous between elements. The interpolation function \(H\) can be written as
\[
H = \begin{bmatrix} H_1 & H_2 & H_3 & H_4 \end{bmatrix}
\]

and
\[
H_1 = \frac{1}{L^3} \left( 2z^3 - 3Lz^2 + L^3 \right) \quad H_2 = \frac{1}{L^2} \left( z^3 - 2Lz^2 + L^3z \right)
\]
\[ H_3 = \frac{1}{L^3}(-2z^3 + 3Lz^2) \quad H_4 = \frac{1}{L^2}(z^3 - Lz^2) \].

Twice differentiating \( \mathbf{H} \) with respect to \( z \) gives
\[ \mathbf{B} = \frac{d^2 \mathbf{H}}{dz^2} = \begin{bmatrix} B_1 & B_2 & B_3 & B_4 \end{bmatrix} \]

and
\[ B_1 = -\frac{1}{L^3}(12z - 6L) \quad B_2 = -\frac{1}{L^3}(6Lz - 4L^2) \]
\[ B_3 = -\frac{1}{L^3}(12z + 6L) \quad B_4 = -\frac{1}{L^3}(6Lz - 2L^2) \].

Let \( r = z/L \), \( \mathbf{B} \) can be expressed in unit coordinates as
\[ \mathbf{B} = \frac{\mathbf{b}}{L^2} \]

where
\[ \mathbf{b} = \begin{bmatrix} b_1 & b_2 & b_3 & b_4 \end{bmatrix} \]

and
\[ b_1 = 12r - 6 \quad b_2 = (6r - 4) L \quad b_3 = -12r + 6 \quad b_4 = (6r - 2)L \].

Then Eqns. (2.20.a,b,c) can be rewritten as
\[ K_1^e = \frac{1}{L^4} \int_L b^e^T \bar{E} \bar{I}_y b^e \, dz \]
\[ K_2^e = \frac{1}{L^4} \int_L b^e^T \bar{E} \bar{I}_x b^e \, dz \]
\[ K_3^e = \frac{1}{L^4} \int_L b^e^T \bar{E} \bar{I}_{xy} b^e \, dz \].

These integrals are carried out using three Gauss points along the length of the member.
2.3.4 Assembly of Element Stiffness Matrix $K^e$

Referring to Figure 2.1, the axial forces are $S_3, S_9$; the torques about the $z_l$ axis are $S_6, S_{12}$; the forces and moments on $xz$ plane are $S_1, S_5, S_7, S_{11}$; the forces and moments on $yz$ plane are $S_2, S_4, S_8, S_{10}$. Combining Eqns. (2.3), (2.17), and (2.20) and rearranging with respect to the corresponding forces and moments give the element stiffness matrix of a 3D beam element:

$$K^e = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix}$$

The expressions for $K_{11}$, $K_{21}$, and $K_{22}$ are in Figure 2.5 and

$$K_{12} = K_{21}^T$$

2.3.5 Transformation of Element Stiffness Matrix

Local element stiffness matrices $K^f_l$ are transformed into global coordinates ($K^e_g$) then assembled into the system stiffness matrix. The transformation matrix $Q^e$ is a $12\times12$ matrix

$$Q^e = \begin{bmatrix} R^e_\xi & 0 & 0 & 0 \\ 0 & R^e_\eta & 0 & 0 \\ 0 & 0 & R^e_\zeta & 0 \\ 0 & 0 & 0 & R^e_\zeta \end{bmatrix}$$

The relation between $K^f_l$ and $K^e_g$ is

$$K^e_g = Q^e^T K^f_l Q^e$$

The transformation matrix $R^e_\zeta$ is defined in the next section.

2.3.6 Reaction and Stress Recovery

After the nodal displacements have been solved, the nodal reactions and stresses can be recovered through postprocessing. When the element stiffness matrix for each
\[ K_{11} = \frac{1}{L^4} \int_L \begin{bmatrix}
  b_1 EI_y b_1 & b_1 EI_x b_1 & L^2 EA & \text{symmetric} \\
  -b_1 EI_{xy} b_1 & 0 & 0 & b_2 EI_y b_2 \\
  -b_2 EI_{xy} b_1 & 0 & 0 & b_2 EI_y b_2 \\
  b_2 EI_y b_1 & -b_2 EI_{xy} b_1 & 0 & 0 \\
  0 & 0 & 0 & L^2 \bar{G} J
\end{bmatrix} \, dz \]

\[ K_{21} = \frac{1}{L^4} \int_L \begin{bmatrix}
  b_4 EI_y b_1 & -b_3 EI_{xy} b_1 & 0 & -b_3 EI_{xy} b_2 & b_3 EI_y b_2 & 0 \\
  -b_3 EI_{xy} b_1 & b_3 EI_x b_1 & 0 & -b_3 EI_x b_2 & -b_3 EI_{xy} b_2 & 0 \\
  0 & 0 & -L^2 \bar{E} A & 0 & 0 & 0 \\
  -b_4 EI_{xy} b_1 & -b_4 EI_x b_1 & 0 & b_4 EI_x b_2 & -b_4 EI_{xy} b_2 & 0 \\
  b_4 EI_y b_1 & -b_4 EI_{xy} b_1 & 0 & -b_4 EI_{xy} b_2 & b_4 EI_y b_2 & 0 \\
  0 & 0 & 0 & 0 & -L^2 \bar{G} J
\end{bmatrix} \, dz \]

\[ K_{22} = \frac{1}{L^4} \int_L \begin{bmatrix}
  b_3 EI_y b_3 & -b_3 EI_{xy} b_3 & b_3 EI_x b_3 & L^2 \bar{E} A & \text{symmetric} \\
  -b_3 EI_{xy} b_3 & 0 & 0 & b_4 EI_x b_3 & 0 \\
  -b_4 EI_{xy} b_3 & -b_4 EI_x b_3 & 0 & b_4 EI_x b_4 & 0 \\
  b_4 EI_y b_3 & -b_4 EI_{xy} b_3 & 0 & -b_4 EI_{xy} b_3 & b_4 EI_y b_4 \\
  0 & 0 & 0 & 0 & L^2 \bar{G} J
\end{bmatrix} \, dz \]

**Figure 2.5** Submatrices $K_{11}$, $K_{21}$, and $K_{22}$ of the stiffness matrix.
element has been formed, it is written to a file for later use in postprocessing. Once
the nodal displacement vector \( \mathbf{d}^e \) is known, the nodal reaction vector \( \mathbf{F}_r^e \) can be
obtained as

\[
\mathbf{F}_r^e = \mathbf{K}^e \mathbf{d}^e - \mathbf{P}^e
\]

where \( \mathbf{P}^e \) is the initial forcing vector. After the moments are computed in \( \mathbf{F}_r^e \), the
standard unsymmetrical bending equation

\[
\sigma_z = \frac{\left( M_x I_y + M_y I_{xy} \right) y - \left( M_x I_{xy} + M_y I_x \right) x}{I_x I_y - I_{xy}^2}
\]

is used to recover the normal bending stress on the cross-section. The axial normal stresses are recovered by averaging the normal force component over the cross-sectional area. The transverse shear stresses are recovered by averaging the shear forces in the two orthogonal transverse directions over the cross-sectional area (\( \tau_{xz} \approx F_x/A, \tau_{yz} \approx F_y/A \)). The shear stresses due to torsional effect are recovered by using Eqn. (2.6) and the shear flow definition \( q = \tau t \).

2.4 Transformation of Coordinates

Local coordinate system is related to the global coordinate system by a coordinate
transformation matrix. This information is needed when element stiffness matrices
are assembled into the system stiffness matrix and the transformation of the mo-
ment of inertia tensor from global coordinates to local coordinates. The coordinate
transformation matrix is derived in the followings.

In Figure 2.6, \( i \) and \( j \) are the two nodes of the beam element. \( x_g, y_g, \) and \( z_g \)
denotes the global coordinates and \( x_i, y_i, \) and \( z_i \) denotes the local coordinates. Let
the length of the element be \( L \) and \( C_x, C_y, \) and \( C_z \) be the direction cosines of \( z_i \) with
respect to the global axes. Therefore,

\[
L = \sqrt{(x_j - x_i)^2 + (y_j - y_i)^2 + (z_j - z_i)^2}
\]
Figure 2.6 Transformation between local axes and global axes.
and
\[ C_x = \frac{x_j - x_i}{L}, \quad C_y = \frac{y_j - y_i}{L}, \quad C_z = \frac{z_j - z_i}{L}. \]

The rotation matrix \( \mathbf{R} \) is found by successive rotations of axes. The transformation from the global axes to the local axes may be considered to take place in two steps. The first rotation is through an angle \( \alpha \) about the \( x_g \) axis. This rotation places the \( z \) axis in the position denoted as \( z_\alpha \), which is the intersection of the \( y_g-z_g \) plane and the \( z_\alpha-x_g \) plane. The second step is a rotation through an angle \( \beta \) about the \( y_\alpha \) axis. This rotation put the \( x_l \) and \( z_l \) axes in their final positions. That is, it places \( x_l \) and \( y_l \) in the same plane as the principal axes of the bending inertia tensor.

Consider first the rotation about the \( x_g \) axis through an angle \( \alpha \). The \( 3 \times 3 \) rotation matrix \( \mathbf{R}_\alpha \) for this transformation is
\[
\mathbf{R}_\alpha = \begin{bmatrix}
1 & 0 & 0 \\
0 & \cos \alpha & \sin \alpha \\
0 & -\sin \alpha & \cos \alpha \\
\end{bmatrix}
\] (2.22)

where
\[
\cos \alpha = \frac{C_z}{C_{yz}}, \quad \sin \alpha = \frac{C_y}{C_{yz}}, \quad C_{yz} = \sqrt{C_y^2 + C_z^2}.
\]

Next, rotate \( z_\alpha \) and \( x_g \) about \( y_\alpha \) by an angle \( \beta \) so that \( z_\beta \) coincides with \( z_l \). The transformation matrix for the second rotation is
\[
\mathbf{R}_\beta = \begin{bmatrix}
\cos \beta & 0 & \sin \beta \\
0 & 1 & 0 \\
-\sin \beta & 0 & \cos \beta \\
\end{bmatrix}
\] (2.23)

where
\[
\cos \beta = C_{yz}, \quad \sin \beta = C_x.
\]

At this point, \( x_\beta \) and \( y_\beta \) are the two orthogonal axes on the plane perpendicular to the local beam axis \( z_l \) but they are not necessarily the principal axes on the plane. If the principal axes are not required, the transformation matrix \( \mathbf{R}_T \) from global system to local system is obtained by multiplying Eqns. (2.23) and (2.22), which yields
\[ \mathbf{R}_T = \mathbf{R}_\beta \mathbf{R}_\alpha = \begin{bmatrix} C_{yz} & -\frac{C_{x}C_{y}}{C_{z}} & -\frac{C_{x}C_{z}}{C_{y}} \\ \frac{C_{yz}}{C_{z}} & C_{x} & \frac{C_{yz}}{C_{y}} \\ \frac{C_{yz}}{C_{z}} & \frac{C_{yz}}{C_{y}} & C_{z} \end{bmatrix} \]  \hspace{1cm} (2.24)

If the local principal axes are known, rotate \( x_\beta \) and \( y_\beta \) about \( z_l \) by an angle \( \gamma \) such that \( x_\gamma \) and \( y_\gamma \) coincide with the local principal axes. The transformation matrix \( \mathbf{R}_\gamma \) for this rotation is

\[ \mathbf{R}_\gamma = \begin{bmatrix} \cos \gamma & \sin \gamma & 0 \\ -\sin \gamma & \cos \gamma & 0 \\ 0 & 0 & 1 \end{bmatrix} \]  \hspace{1cm} (2.25)

The total transformation matrix \( \mathbf{R}_T \) is obtained by multiplying Eqns. (2.25) and (2.24):

\[ \mathbf{R}_T = \mathbf{R}_\gamma \mathbf{R}_\beta \mathbf{R}_\alpha = \begin{bmatrix} C_{yz}\cos \gamma & -\frac{C_{x}C_{y}\cos \gamma + C_{z}\sin \gamma}{C_{y}} & -\frac{C_{x}C_{z}\cos \gamma - C_{y}\sin \gamma}{C_{y}} \\ -\frac{C_{yz}}{C_{z}}\cos \gamma & C_{x} & \frac{C_{yz}}{C_{y}} \frac{C_{yz}}{C_{y}} \\ C_{x} & \frac{C_{yz}}{C_{y}} & C_{z} \end{bmatrix} \]  \hspace{1cm} (2.26)

### 2.5 Model Verification

The element stiffness matrix and related transformations have been implemented in a FORTRAN subroutine and combined with the finite element program MODEL developed by Akin [3]. The beam model are verified for its validity and accuracy by five examples.

#### 2.5.1 Bending of a Rectangular Cantilever Beam

The first test is the accuracy of the beam model in predicting the bending stress of a constant cross-sectioned rectangular cantilever beam. The beam is 15.8 cm long and the cross-sectional dimension is 2 cm by 4 cm. The dimension of the beam is illustrated in Figure 2.7.
A vertical load of 797 N is applied at the tip of the beam. The beam is divided into 23 beam elements. The bending moment and bending stress are presented in Figure 2.8. The numerical results are compared with analytical solutions. The beam model results agreed with the analytical solution to six significant digits.

2.5.2 Torsion of a Tapered Thin-Walled Cantilever Beam

The second test is the accuracy of the beam model in predicting angle of twist of a tapered thin-walled cantilever beam. The thickness of the wall is 0.2 in. The length of the beam is 10 in. The diameters on the two ends of the beam are 1 in and 2 in respectively (Figure 2.9(a)). The shear modulus of the material is 12,000,000 psi. A torsional load of 50,000 in-lb about the long axis of the beam is applied at the free end. The exact solution of angle of twist, \( \phi \), can be calculated by [23]

\[
\phi = \frac{2 \cdot TL \cdot (d_a + d_b)}{\pi G t \cdot d_a^2 \cdot d_b^2}
\]

where \( T \) is the torque, \( L \) is the length of the beam, \( G \) is the shear modulus, and \( t \) is the thickness of the wall. The beam model uses 20 elements to approximate the beam. The beam model results are compared with the analytical solution in Figure 2.9(b). The beam model results agreed with the analytical solution with an error less than 0.1%.

2.5.3 Unsymmetrical Bending of a Z-shaped Cantilever Beam

The third example is a straight, constant cross-sectioned Z-shaped cantilever beam with an inclined load at the free end [9] (Figure 2.10). This is an unsymmetrical bending case to test the model in predicting unsymmetrical bending moment and stresses. The maximum bending stress in the cross-section is predicted by the beam model with an error less than .1%.
Figure 2.8  Bending moment and bending stress in a rectangular cantilever beam.
Figure 2.9 Dimension and angle of twist of a thin-walled cantilever beam.
Figure 2.10  Cross-section of a Z-shaped cantilever beam.
2.5.4 Bending of a Tapered, Variable Cross-sectioned Cantilever Beam

The fourth example is to test the beam model in predicting bending moment and stress in a variable cross-sectioned cantilever beam [4]. The dimension of the beam is illustrated in Figure 2.11. The results from the beam model is compared with the results from Advanced Continuous Simulation Language (ACSL) [2]. The two numerical results have a difference less than 1%. The bending stresses in the upper fiber of the beam from two numerical calculations are shown in Figure 2.12.

2.5.5 Bending and Torsion of a Composite Beam

The fifth example is to test the beam model in predicting bending stresses and torsional shear stresses in a composite cantilever beam with circular cross-sections (Figure 2.13). This structure is a simplification of an intact long bone. The outer thin-walled tube is cortical bone and the inner core is trabecular bone. The Young’s moduli for cortical bone and trabecular bone are 14 GPa and .5 GPa, respectively. The beam is first subjected to a vertical load of 100 N. In one case, the beam is simulated as one composite beam and in another case it is simulated as two separate beams overlapped together - a beam of cortical bone and a beam of trabecular bone. Stresses predicted by the two cases are the same. The displacement and bending stress on the outer fiber of the beam under bending load predicted by the beam model and the analytical solution are shown in Figure 2.14. The beam results have an error less than .5% compared to the analytical solution.

The composite beam is also subjected to a torsional load of 100 N-mm about its long axis. When simulating the composite beam as two separate beams, the torques carried by the two materials are proportional to their torsional rigidity ($GJ$) ratio. The angle of twist and torsional shear stress predicted by the beam model have an error less than .1% when comparing to the analytical solution.
Figure 2.11 Dimension and load of a variable cross-sectioned cantilever beam.
Figure 2.12 Comparison of bending stresses from beam model and ACSL.
Figure 2.13 Dimension and loads of a composite cantilever beam.
Figure 2.14 Comparison of displacement and bending stresses in the composite beam.
2.6 Summary

An unsymmetric FE beam model is developed in this chapter. The beam model is unique in coupling the bending effects in two orthogonal transverse directions and calculating torsional shear by using the theory of closed thin-walled sections. The cross-sectional properties are calculated from CT scan data and variable properties along the long axis of the beam are used. The model is verified for its validity and accuracy by five test examples and the results are in good agreement with the analytical results or results from other numerical methods. The accuracy of the beam model in predicting stresses in intact femur and femur with a prosthesis will be presented in the later chapters.
Chapter 3

Three-Dimensional Stress Analysis of an Intact Femur

3.1 Introduction

Understanding of the stresses in intact bones has been a longstanding interest since mid-19th century. The early works by Meyer [41], Wolff [65] and Koch [37], although were not done with finite element, provide a certain amount of motivation for finite element studies of the femur. Correct prediction of the stress state of the femur is important for the design of femoral prosthetic devices. Many works have been done, either using finite element or experimental methods, to compare the stress state in the femur before and after total hip replacement to evaluate the performance of a certain type of prosthesis. Several computational bone remodeling models uses the stress or strain state in the intact femur as the base reference for a stimulus of the remodeling process. Correct modeling of the femur involves good approximation of the complex geometry and its representation in the finite element model, reasonable representation of the material properties of cortical and trabecular bones and appropriate loading and boundary conditions to model everyday activities. All the above parameters will affect the validity of the finite element model. Model verification is of paramount importance before the finite element model results can be applied to the problems concerned.

In this chapter, a brief literature review is first presented. Then the method to construct an anatomically realistic model of the proximal portion of a femur is described. The stress analysis results are then compared with experimental data from the literature and the accuracy is also checked by an error estimator. Results
from the beam model proposed in the previous chapter are also compared with the 3D solid model.

3.2 Literature Review

Three-dimensional finite element models of the intact femur have been done by many researchers since the method was first introduced to the orthopedic research in the early 1970's. Works before 1982 have been summarized by Huiskes and Chao [30] and Rybicki [55]. Models in the early years were focusing on the use of the finite element rather than solving a specific problem. All these models used the "conventional method", i.e. to section an intact femur or use the x-ray of the cross-sections of the femur, then hand digitized the contours of the cross-sections and construct the geometry by stacking up these contours. Mesh generation was all done by hand. The difficulties of handling the large amount of data generated by a 3D model and the time and complexity shown in these models have precluded the routine use of these methods.

Harrigan et al. [26] developed a linear femur-implant FE model to study the initiation of failure of fixation in cemented femoral total hip components. There was a problem of consistency of the location of the origin of each cross-section which resulted in a unrealistic rugged cortex surface and implant-cement interface. This will very clearly influence the interface stress estimates. This problem of uncertainty of origin will be eliminated when CT scans are used for the generation of geometric model. In their model, linearly interpolated element and linear material properties were used. Vichnin and Batterman [62] developed a linear model of proximal femur with and without implant to predict the failure loads for the implant. In their model, both isotropic and transversely isotropic material properties were employed in the diaphysial cortex. Lotz et al. [39, 40] used both linear and non-linear material
properties for the bone on a FE model to do fracture prediction for the proximal femur. Cheal et al. [17] developed a 3D FE model with and without an implant to study the influence of several different loading cases and prosthetic material properties on the mechanics of the proximal femur. Rigid bonding of the implant-cement interface was assumed and conventional model building methods were used for the above models.

Keyak [34], and Chang [16] have developed automated methods to create FE model directly from CT scan data using linearly interpolated hexahedral elements. The advantages of these methods are that human intervention is minimized in the FE model building and anisotropic material properties of the bone can be constructed directly from the CT density data at each pixel. The methods will generate a large amount of nodes and elements which is costly in terms of CPU time to solve the large system of equations. Also the cortex surface and bone-implant or implant-cement interface are not smooth from these methods which makes the modeling of non-linear interface difficult. Kim et al. [36] has made a modification to Keyak’s method [34] that used wedge elements to produce a smoother representation of the bone surfaces. Keyak’s group also suggested that in order to get accurate stress/strain results using their automated method, the element size must be very small (3 mm on one side or smaller) [35]. Due to the sensitivity of stresses to element size in their model, they suggested that quantitative comparisons of the stress/strains are meaningful only when the same size of elements are used in different models.

Strain data for the intact bone have been measured experimentally before and after total hip arthroplasty by using strain gages [19, 44, 45]. These data are valuable for verification of finite element models.
3.3 Methods

A method based on the analytic solid modeling (ASM) methodology is used in this research to build anatomically correct FE models. An overview of the ASM methodology is presented in [15]. This method utilizes CT scan data and the commercial computer-aided engineering (CAE) software PATRAN (PDA Engineering, Costa Mesa, California) to construct the finite element model. The CT scan provides geometric data up to 98% accuracy [59]. This is desirable when constructing an anatomically realistic model. An intact femur model is presented in this chapter and a model with an implant will be presented in chapter 5. A flow chart of the general procedure is presented in Figure 3.1. The description of each step follows.

3.3.1 Geometric Model

CT Image Acquisition To accurately model the complex geometry of the femur noninvasively, cross-sections of CT scan of the proximal part of an intact right femur are used to reconstruct the geometry of the model. The scanning was done with 3 mm interval between every two slices from top of the femoral head down to the lesser trochanter. In the region distal to the lesser trochanter, a 5 mm interval was used. There are 64 slices of CT scan for a total length of 287 mm along the long axis of the femur.

The CT scan coordinates are adopted as the global coordinates for the FE model. The origin of the global coordinates is the CT scan origin on the most distal slice. The x axis is from lateral to medial. The y axis is from anterior to posterior. The z axis is from distal to proximal. Therefore, the xz plane is parallel to the frontal plane; the xy plane is parallel to the transverse plane; and the yz plane is parallel to the sagittal plane. The anatomic directions of the model is illustrated in Figure 3.2.
Figure 3.1 Flow chart of the general procedure to construct the 3D solid model.
Figure 3.2 Anatomic directions of a right intact femur.
Automatic Contour Detection  Inner and outer contour data were generated directly from CT scan data by an in-house routine [16]. Not all CT slices are used in building the geometric model.

Grids Selection on Contours  On each CT slice, a few characteristic grids are selected from the inner and outer contour of the cortical bone. These grids are selected so that the geometric shapes of the original contours can be properly represented by piecewise parametric cubic curves based on ASM theory [15]. These grids are picked by human judgement. The topology of the grids should remain the same while the coordinates change slightly from patient to patient. If these grids can be picked automatically by a computer algorithm then the construction process of the 3D solid model can be fully automated.

Generation of Lines, Patches, and Hyperpatches  After these characteristic grids have been selected, the line generation functions in PATRAN were used to create parametric cubic curves from these grids. From the parametric curves, parametric bicubic patches are generated using the patch generation functions. Finally, parametric 3D solids (hyperpatches) are generated from patches. These hyperpatches (HPAT) are the building blocks for FE mesh generation. There are 62 HPAT’s and 199 lines generated in our typical femur model. Parametric lines and a set of well connected parametric HPAT’s for the model are illustrated in Figure 3.3. Once these HPAT’s are defined mathematically, FE mesh generation can be performed in these HPAT’s one by one.

3.3.2 Finite Element Modeling

Mesh Generation  Mesh generation of the model is done individually in each hyperpatch. Element numbers in the three local parametric directions of each hyper-
Figure 3.3  Parametric lines and hidden line plot of the geometric model of the intact femur.
patch are defined separately. Transitional elements are automatically generated when different number of elements are specified on opposite side of the same parametric direction.

The type of element used in each hyperpatch is also assigned in this step. The element types used in this linear model are 8-node tri-linear hexahedral elements and 6-node wedge elements in the proximal region where thick layer of cortical becomes thin. In the most proximal part of the femur, 4-node thin shell elements were used to account for the thin cortical shell. 0.1 mm thickness is assigned to all shell elements in this region.

The number of elements specified in each parametric direction must be equal for all hyperpatches sharing the same edge to assure mesh compatibility between adjacent hyperpatches. The finite element model has 260 and 384 solid elements for cortical and trabecular bones, respectively and 194 shell elements for the thin layer of cortical bone. The element groups for cortical bone, trabecular bone, and thin layer of cortical bone are illustrated in Figure 3.4.

**Node Equivalencing and Model Optimization** When elements are generated in each hyperpatch, nodes related to each element are also generated. Since element meshes are generated in each hyperpatch, duplicated nodes are generated on the faces which are shared by two adjacent hyperpatches. These duplicated nodes are equaled in this step and the lower number is kept after equivalencing.

After nodes are equaled, the whole model can be optimized by compacting the node numbers. The node numbers can also be resequenced according to wavefront or bandwidth optimization. There are a total of 865 nodes in the proximal femur model (Figure 3.4).
Figure 3.4  Element groups for (a) solid cortical bone (b) solid trabecular bone and (c) thin shell cortical bone.
Assign Material properties, Loads, and Boundary Condition  The material properties of the model are assumed to be isotropic for both cortical bone and trabecular bone. The Young's modulus for the cortical and trabecular bones are 14 GPa and .5 GPa respectively. Poisson's ratio is assumed to be 0.3 for both materials.

In order to compare the results from the 3D model with published experimental results [45], the same loading conditions as the experiment are used for the 3D model. The first case is an axial load of 2000 N through the center of the femoral head in the frontal plane inclined by 15 degrees from the vertical (z) axis. The second case is a torsional load of 20 N-m about the vertical axis through the center of the femoral head.

The boundary conditions for this model is assumed that the distal end of the femur is fully clamped. All degrees of freedom are constrained for all nodes on the bottom face of the distal end.

Neutral File Generation  After the above necessary data are assigned in the FE model, a neutral file can be generated containing all geometric and analytic model informations. The neutral file system is device independent and can be translated into most commercial computer-aided design (CAD) and finite element analysis packages by an appropriate interface translation software.

Transport to FE Package  The FE package chosen for the finite element analysis in this research is ABAQUS [1] due to its availability and capability of solving non-linear interface problem. The neutral file is translated into ABAQUS input file by the PATRAN-ABAQUS interface software PATABA. Another option for the FEA package is P/FEA which is directly connected to PATRAN. The FE model is solved on a SUN SPARC-2 machine. The advantage of using P/FEA is the availability of
an error estimator which is very useful in evaluating the accuracy of the FE model. This will be discussed in the result and discussion section.

**Results Translation back to PATRAN** After the FE model is solved by ABAQUS, the nodal displacements, stresses and strains are translated back to neutral file format by the interface software ABAPAT. This result neutral file can be read by PATRAN for result evaluation.

**Results Evaluation** The enormous result output from a 3D finite element model can be effectively evaluated using PATRAN. Color contour or fringe plots can be generated to visualize the results.

### 3.4 Model Verification

In order to verify the accuracy of the model, the FE results are first compared with published experimental data and then checked by an error estimator.

#### 3.4.1 Comparison with Experimental Results

The FE strain results are compared with published strain gage experimental data [45]. There were two load cases applied in the experiment. The first one is an 2000 N axial load inclined medially by 15 degrees from the vertical in the frontal plane (Figure 3.5a). The second load case is a 20 N-m torsional load about an axis passing through the femoral head and parallel to the long axis of the femur in the transverse plane (Figure 3.5b).

A 16 mm long prosthesis was also used in the experiment for comparison before and after implantation. The strain gage in the experiment are placed both medially and laterally at four different levels along the long axis of the femur. The four levels were: No. 1, 10 mm below the femoral neck osteotomy; No. 2, midway down the
Figure 3.5  Loading conditions for (a) inclined axial load and (b) torsional load.
medial cortex between No. 1 and 3; No. 3, the level of the stem tip; and No. 4, 20 mm below No. 3. A schematic illustration of these four levels is shown in Figure 3.6. Longitudinal and circumferential strain values from the FE model corresponding to these four levels both medially and laterally are compared with the strain gage measurements.

The comparison under inclined axial load is presented in Figure 3.7. The FE results showed a similar trend to the strain gage results at all levels. Under axial load, the longitudinal strains decrease from proximal to distal with the highest values in the calcar region. Lateral tensile strains are much lower than the compressive strains on the medial side of the femur. Circumferential strains are much lower than longitudinal strains. The largest difference in value between FE results and strain gage results is the longitudinal strain in the calcar region. In another published experimental data [44], under the same loading condition, the average longitudinal strain was 1827 $\mu\varepsilon$ (micro-strain) which is closer to the FE result than [45].

Under torsional load (see Figure 3.8), FE results also show same trend as the experimental results except for the longitudinal strains on the lateral side. FE results showed much lower longitudinal strain values than the strain gage measurement. It is suspected that in that experiment, there was a vertical load component involved in addition to pure torsion in the transverse plane. Therefore, a vertical load of 80 N applied at the top of the femoral head was added in the FE model in addition to the torsional load to test this observation. The new results (Figure 3.9) showed a closer trend with the experimental results than pure torsion load case. But there are still some discrepancies in the longitudinal strains on the lateral side. It is concluded that there are some other unknown load conditions involved in the experimental setup.

Under torsional load, both longitudinal and circumferential strains are much lower than those under axial load. The proximal part of the femur is often exposed to
Figure 3.6  Strain gage levels on the intact femur model
Figure 3.7 Comparison of strain values between FE model and strain gage under inclined axial load.
Figure 3.8 Comparison of strain values between FE model and strain gage under torsional load.
Figure 3.9 Comparison of strain values between FE model and strain gage under combined torsional load and vertical load.
high longitudinal strains and the bone structure of this region is equipped for high longitudinal deflection.

Although the general trends of strain distribution in different femora are the same, there are large differences in magnitude in the individual femur as seen in the published experimental data [44]. This implies that in order to compare stress/strain values before and after total hip replacement, each femur should be its own control. The similar trend in the strain distribution under either axial or torsional load showed that the FE results are qualitatively in good agreement with the published experimental results.

3.4.2 Error Estimate

The error estimate in PATRAN is essentially based on the differences between the averaged and the unaveraged element stresses. If these differences vanish, the FEA results will be exact. Let \( \sigma \) be the unaveraged element stresses, \( \sigma^* \) be the averaged element stresses, and \( e_\sigma = \sigma^* - \sigma \) be the error in stress. The following error measure [66], based on the energy norm, to compute one error number:

\[
\| e_\sigma \|^2 = \int_{\Omega} e_\sigma^T C^{-1} e_\sigma \, d\Omega
\]

where \( C \) is the material stiffness matrix. The above error measure can be interpreted as a weighted root mean square (RMS) error in the stresses. The error is computed as

\[
\eta' = \frac{\| e_\sigma \|}{\sqrt{U^2 + \| e_\sigma \|^2}}
\]

where \( U \) is the total strain energy of the model. Using the above equation, the error estimate for each element is computed as

\[
\eta' = \frac{\| e_{\sigma_i} \| / V_i}{\text{Strain energy for the whole model}}
\]
where $\|e_{\sigma}\|$ is the weighted RMS error in stresses in the i-th element and $V_i$ is the element volume.

The error estimate for the intact femur model under a single-leg stance load is shown in Figure 3.10. The average error for the whole model is less than 1%. The highest error is 3.6% occurred at the neck region where the geometry of the femur changes abruptly.

### 3.5 Comparison with Beam Model Results

The intact femur used in this chapter has also been analyzed by the beam model proposed in the previous chapter. The load case applied to the FE model simulates a single-leg stance which is a common daily activity.

#### 3.5.1 Load Calculation During Single-Leg Stance

In calculating the loads acting on femur during single-leg stance, the simplified free body technique [22] is adopted. It is assumed that there are three major forces acting on the femur during single-leg stance (see Figure 3.11a): the ground reaction force $W$ against the foot, which is transmitted through the tibia to the femoral condyles; the abductor muscle force $M$ produced by contraction of the abductor muscles; and the joint reaction force $J$ on the head of the femur. These three forces are assumed to be coplanar and acting in the frontal plane. All other muscle forces are neglected. The characteristics of these three forces are:

- The ground reaction force $W$ has a known magnitude equal to five-sixths of body weight (BW) and a know sense (pointing upward vertically).

- The abductor muscle force $M$ has a known sense and direction (20 degrees from vertical [32], inclined medially). Its magnitude is to be determined.
Figure 3.10  Error estimate of the intact femur model under single-leg stance load.
• The joint reaction force \( J \) has a known point of application on the surface of the femoral head. Its direction and magnitude are determined from experimental data from Davy et al. [20]. It is 16.6 degrees from vertical. Its average magnitude is 2.1 BW.

These three forces are in equilibrium and a triangle of forces in equilibrium is constructed in Figure 3.11b. Solving for the unknowns graphically or algebraically, the magnitude of the abductor force is found to be 1.754 BW and the joint reaction force is inclined medially by 16.6 degrees. Assuming the body weight of the patient to be 696.5 N (or 71 Kg, 156 lbs), the components of joint reaction force in the model coordinates are -419 N in \( x \) and -1407 N in \( z \); the components of the abductor muscle force are 418 N in \( x \) and 1148 N in \( z \).

3.5.2 Beam Model of an Intact Femur

There are 64 CT slices along the long axis of the femur. Each slice is used as a node in the beam model and the node's location is at the elastic centroid of the cross-section. There are 64 nodes and 63 beam elements in the beam model. Cross-sectional properties at the center of each beam element are calculated by cubic interpolation using all slices. Stresses are post-processed after the displacements of each node is solved. Same material properties for cortical and trabecular bone are used in the beam models as in the 3D model. The boundary conditions for beam model is a fixing of the most distal node of the model. The joint contact force is applied to the most proximal node. Since the point of application of the abductor muscle force is not one of the nodes in the beam model, it is transferred to an interior node on the beam with statically equivalent force and couple. The beam elements for the intact femur model are plotted together with each cross-sectional contours in Figure 3.12.
3.5.3 Results and Discussion

The von Mises stresses on the medial and lateral cortex of the femur for the two models are presented in Figure 3.13. The von Mises stresses in the diaphysial region of the femur predicted by the beam model are in close agreement with those of the 3D solid model. In the region proximal to the lesser trochanter where the femoral geometry varies a lot, the stresses predicted by the beam model show a similar trend as the 3D solid model but the differences in value are up to 30%. This is similar to the conclusions from Rybicki's paper [54]. This implies that beam theory is appropriate for the calculation of stresses in the diaphysial region of the femur. While in the region proximal to the lesser trochanter, a continuum theory is required. An anatomic 3D solid model is needed to accurately predict the stresses in this region.

The large difference at the distal end of the model is due to the difference in defining the boundary conditions. In 3D solid model, all nodes on the most distal
Figure 3.12  Beam elements and cross-sectional contours for the intact femur model.
cross-section are constrained, while in beam model only the centroid of the cross-section is fixed.

The beam model requires much less time in data preparation and very short CPU time to solve comparing to 3D solid model (less than 2 seconds to 6 minutes ). It is advantageous to use the beam model to do parametric studies of the preliminary design of a prosthesis.

3.6 Summary

A method to construct an anatomically correct 3D solid finite element model of an intact femur from CT scan data is developed in this chapter. Stresses/strains predicted by the 3D model are compared with experimental data and the accuracy is checked by an error estimator and compared to a new beam model in 3D space that includes unsymmetrical bending and the torsion of thin-walled closed sections. The results showed that the model constructed by this method is reliable in assessing stress/strain in the femur.

The advantages of using this method to construct 3D finite element solid models are:

- Models constructed by this method predicts stresses/strains accurately with much fewer numbers of nodes and elements than models constructed by other automatic methods such as octrees [16, 34]. This will be economical of both data preparation and CPU time.

- The geometry of the model is modeled accurately and the outer surfaces and boundaries between two different materials are smooth. This is desirable in modeling the non-linear behavior of the bone-implant interface. Study of relative motion are possible and implant stability can be determined. These will be further demonstrated in later chapters.
Figure 3.13  Comparison of von Mises stresses on medial and lateral cortex of the intact femur.
• The mesh density of the models can be locally adjusted as desired within each hyperpatch.

• The model is transportable to most finite element and CAD software and any in-house finite element codes through a neutral file.

The main disadvantage of this method is that the model is not fully automated. Human intervention and judgement are still needed in the model building process, especially in selecting the characteristic grids from contours. If that can be done automatically, there is a greater potential for this method to relief the burden of manual input.
Chapter 4

Non-linear Interface Analysis of an Idealized Symmetric Bone-Implant Model

4.1 Introduction

Aseptic loosening is by far the most frequent long-term complication in total hip arthroplasty (THA). It is the limiting factor for the functional life span of the bone-implant structure. Many technological improvements have been introduced to prolong clinically acceptable prosthesis function. These efforts were predominantly directed at understanding and postponing aseptic loosening of the fixation.

Aseptic loosening is a gradual process. When the mechanical integrity of the bone-implant interface is lost and a fibrous tissue is formed between the two surfaces, the patient develops pain and functional restrictions. The initial fit and stability of the prosthesis are the most important factors for successful uncemented THA. Relative motions in the bone-implant interface will prevent bone tissue from growing into the porous-coated implant surface. It may also cause bone to resorb at the interface and create a fibrous-tissue membrane. An understanding of the relative motion and stress distribution in the interface is important for prosthesis design.

The finite element method (FEM) has been used to simulate the interface conditions of the bone-implant interface. Most published literature in this area assumed the bone-implant interface condition as either fully bonded or no-tension, no-friction. The interface condition in the early post-operative period is neither fully bonded nor frictionless. Experimental results [58] showed that the interface behavior is non-linear and there is friction behavior involved in the interface and the friction coefficients for different implant surface finishes have been measured. The typical range of the coef-
icient of friction is from 0.3 to 1.3. It is not clear how these assumptions affect the stress distribution in the interface and relative motion between the bone and implant comparing to the case with friction involved. It is desirable to include friction behavior in the modeling of bone-implant system to more accurately approximate the real situation after total hip arthroplasty.

The purposes of this chapter are to study the non-linear behavior of the interface by including interface elements in the finite element (FE) model. The different assumptions for the interface, i.e. fully bonded; no-tension, no-friction; and no-tension with friction will be studied to assess their influences on regions of contact, interface stress distribution, and bone-implant relative motion. A simplified symmetric model will be used to address these issues.

4.2 Literature Review

Study of bone-implant interface behavior has been an important topic in prosthesis design and is gaining more attention by orthopedic researchers. Rohlmann et al. [53] used a geometrically simplified finite element model to study load transfer between a porous-coated hip endoprosthesis and a femur. The assumptions for the interface condition were both rigidly bonded and non-linear interface. They found that the maximum relative motion in the interface occurs for implants with the same elastic modulus as compact bone. The elastic modulus of the porous coating had only a small influence on the bone stress.

Huiskes [31] used a 2D variable thickness side plate FE model to study the general differences in load-transfer mechanisms and stress patterns of cemented, fully ingrown, proximally ingrown, and smooth press-fitted femoral stems in total hip arthroplasty. For press-fit stem and non-coated region of partly coated stem gap elements were included to allow compressive stress transfer only. Slip and tensile separation may
occur owing to the non-linear gap element applied. He found that the load transfer mechanism is similar for all bonded configurations but differs dramatically for unbonded stems. In the press-fit stem, the interface stresses are affected more by stem shape as a geometric entity and less by stem rigidity.

Harrigan and Harris [24] used a straight tubular bone and a metal cylindrical prosthesis to study the contact areas and pressures in the noncemented bone-implant interface. Frictionless 3D gap elements were included in their model. They found that contact between the prosthesis and bone is an essentially 3D effect. Gaps of less than 20 µm between bone and implant can substantially change contact stress distributions.

Harrigan and Harris [25] also used an anatomically realistic 3D FE model to study the effect of debonding in cemented hip prosthesis. Frictionless 3D linear contact elements were included in their model. They found that cement failure subsequent to prosthesis debonding is likely in the proximal region in a partially debonded implant due to stair climbing loads and is likely below the prosthesis tip in a fully debonded implant due to gait loading.

Rancourt et al. [52] and Shirazi-Adl et al. [58] experimentally measured the friction coefficients between different implant surface finishes and trabecular bone. Their results showed that the interface friction curve is highly non-linear. Friction coefficients measured vary between 0.3 and 1.3. The maximum resistance in friction is independent of the bone excision site, type of porous-surfaced metal plate, magnitude of normal load, and placement of bone cubes on metal plates or vice versa. The smooth-surfaced metal plate has significantly smaller friction resistance than porous-coated ones.
4.3 Methods

An idealized bone-implant FE model is constructed using geometric primitives such as cylinders, cones, and elbows. The model is symmetric to the mid-frontal plane ($xz$ plane) of the bone-implant structure. The primitives used to construct the idealized model are demonstrated in Figure 4.1. The dimensions of the model are approximations to the dimensions of an anatomic bone-implant model which will be constructed in the next chapter. The total length of the model is 244 mm and the length of the stem is 155 mm. The stem is slightly tapered and a curvature is present in the proximal region.

The quadratically interpolated interface elements INTER9 from ABAQUS [1] are included to simulate the non-linear behavior of the bone-implant interface. It has 9 nodes on either face of the element. This interface element and its topology are illustrated in Figure 4.2. This element will only transmit compressive normal stresses and shear stresses. When tensile normal stress is present, separation occurs. A friction coefficient can be assigned to the interface element to simulate the non-linear friction and sliding behavior in the interface. The numerical solution process is different from that of a linear model. In the linear model, the system stiffness matrix is solved only once. In the non-linear model, loads are divided into small steps and the numerical solution for the system must converge in a step before the next load step is applied to the model for the next iteration. This process goes on iteratively until the full load is reached. The accuracy and reliability of the interface element in predicting normal and shear stresses are verified by a simple axisymmetric model presented in Appendix A.

Hyperpatches for cortical bone, trabecular bone, metal implant, and interface are generated from geometric primitives. 20-node quadratic hexahedral elements are then generated in each hyperpatches. This kind of element is compatible with the
Figure 4.1  Geometric primitives used in constructing the idealized symmetric bone-implant model.
Figure 4.2  Topology of the interface element INTER9.
quadratic interface element used in the model. The material properties for cortical bone, trabecular bone, and metal implant are 18 GPa, .5 GPa, and 200 GPa respectively. Poisson’s ratio is assumed to be 0.3 for all materials. The loading condition simulates loads generated during single-leg stance.

Due to the symmetry of the model, only half of the model is simulated in the analysis. Appropriate boundary conditions are assigned to the nodes on the symmetry plane (mid-frontal plane). All nodes on the most distal cross-section are fixed. The finite element mesh is generated according to the experience with the axisymmetric model in Appendix A. The average error for the model is less than 1%. There are 68, 48, 84, and 48 elements generated for the cortical bone, trabecular bone, metal stem, and bone-implant interface respectively. The different element groups are shown in Figure 4.3. There are a total of 1450 nodes for the linear model (fully bonded interface) and 1607 nodes for the non-linear model.

Three cases of interface condition are simulated. For the fully bonded case, there is no interface element between trabecular bone and implant. The same node number is assigned at the same location in the interface to the two materials. For the no-tension, no-friction case, interface elements are included. One face of the interface element is connected to the trabecular bone and the other face is connected to the implant. A perfect fit is assumed. Therefore, the corresponding nodes on the two sides of the element have the same global coordinates. The friction coefficient for the interface is assigned 0. No shear stress is transmitted through the interface element. Only compressive normal stresses are transmitted through the interface in this case. For the no-tension, with friction case, a friction coefficient of 0.47 is assigned for the interface element. This friction coefficient is in between those of a smooth surface finish (0.42) and a normal porous-coated finish (0.61) according to Shirazi-Adl’s experimental data [58]. The assigned friction coefficient complys with a friction angle of 25 degrees. In
Figure 4.3 Element groups for the idealized bone-implant model.
this case, both shear stress and compressive normal stress can be transmitted through the interface. When tensile normal stress is present, debonding occurs. The regions of contact, relative motion, and interface stresses in the bone-implant interface will be studied for the three interface condition assumptions.

4.4 Results

Regions of Contact

Since the bone-implant interface will debond at the nodes where tensile normal stress occurs, the contact and separation condition after loading for each interface element can be defined based on the number of nodes being in contact. It is assumed in this study that if there are 7 to 9 pairs of nodes in one interface element being in contact, the two sides of the element is in contact; if there are 4 to 6 pairs being in contact, the contact condition for the element is in transition; if there are 0 to 3 pairs being in contact, the two faces of the element are in separation. According to this definition, the contact regions in the interface for the idealized model after loading are illustrated in Figure 4.4.

Under single-leg stance loading, the frictionless case has larger regions of contact than the friction case. That is especially obvious on the lateral side of the interface. On the medial side, the friction case has a larger separation region near the curve and straight stem junction and at the tip of the stem. For the fully bonded case, the whole bone-implant interface is in contact (not shown in the figure).

Relative Motion

The relative motions between bone and implant are calculated at eight sample nodes on 4 levels on both medial and lateral sides of the interface. The locations of these sample nodes are shown in Figure 4.5. The first level is at the stem tip. The second
Figure 4.4 Regions of contact for frictionless case and friction case.
level is at the middle of the straight section of the stem. The third level is at the junction of the curve and the straight sections of the stem. The fourth level is at the top of the interface. The relative motions are examined by their axial and transverse components and their resultant. The axial relative motion is calculated at a sample node as the difference between the $z$ displacements ($z$ axis is parallel to the long axis of the stem) of the stem node and the bone node. The transverse relative motion is the resultant of the differences in the $x$ and $y$ components of the displacements between the two nodes. The total relative motion is the resultant of the axial and transverse components.

The relative motions for the frictionless case (case 1) and the friction case (case 2) are compared in Figure 4.6. Under single-leg stance loads, the axial relative motions are much larger than transverse relative motions at all sample nodes in both cases. The frictionless case has much higher relative motion than the friction case. On average, the total relative motion predicted by the frictionless case is 8 times of that predicted by the friction case. The maximum relative motion on the medial side occurs at the tip of the stem (sample node 1). On the lateral side, the maximum relative motion occurs at the top of the interface (sample node 4). For the fully bonded case, since same node number is used at the same location of the interface, there is no relative motion between bone and implant.

**Interface Stresses**

The interface normal stresses are compared among the three cases along a line of nodes on both medial and lateral side of the interface and on both bone side and implant side. The normal stresses on the medial and lateral sides of the implant are shown in Figure 4.7. The largest normal stress on the lateral side occurs at the tip of the implant. The frictionless case shows a little higher normal stresses along the most
Figure 4.5 Locations of sample nodes for relative motion calculation on the medial and lateral sides of the interface.
Figure 4.6  Comparison of relative motions for the frictionless case (case 1, \( \mu = 0 \)) and friction case (case 2, \( \mu = 0.47 \)).
part of the interface. There is a sudden peak of normal stress occurs at the junction of the curve and straight sections on the lateral side of the stem and this is not seen for the fully bonded and friction case. On the medial side of the implant, the fully bonded case shows a large tensile normal stress at the stem tip. This tensile normal stress disappears when no-tension interface elements are included in the model as seen in the frictionless and friction cases. The normal stresses on the medial and lateral aspects of the interface on the bone side show a similar pattern of stress profile as on the implant side.

The fully bonded case predicts lower normal stresses on the lateral side and most of the medial side comparing to cases with no-tension interface element. The highest normal stresses predicted in the three cases are at the lateral stem tip.

The von Mises stresses in the trabecular bone for the three cases are shown in Figure 4.8. The frictionless case shows higher von Mises stress than the other two cases. The stress patterns for the bonded and friction cases are similar. The major difference between the two cases is at the medial side near the stem tip where large tensile normal stress is present for the bonded case while the friction case has bone-implant separation.

### 4.5 Discussions

The initial relative motion stability of the uncemented implant in the host bone is the most important factor to ensure the success of the operation. To model the complex bone-implant system, the bone-implant interface condition is an important parameter to be considered. How the interface is modeled will influence the stress distribution and interface behavior.

For the fully bonded assumption, the relative motion between the bone and implant can not be modeled since they have the same displacement everywhere in the
Figure 4.7  Normal stresses on medial and lateral sides of the implant.
Figure 4.8  Von Mises stresses in trabacular bone for the three interface conditions.
interface. Under single-leg stance loading, this assumption predicts high tensile normal stresses near the medial stem tip. This is not realistic for the early post-operative period since there is no strong bonding between bone and implant exists at this time.

For the no-tension, no-friction assumption, since no shear stress is transmitted, the implant subsides much more than the friction case to generate large compressive normal stresses across the interface in order to transfer load from stem to bone. This also causes larger regions of contact than friction case and higher von Mises stresses in the surrounding trabecular bone comparing to another two cases. The frictionless assumption predicts unrealistically large relative motion in the interface. The large relative motion might inhibit bony ingrowth or osseous integration.

4.6 Summary and Conclusion

A simplified bone-implant model symmetric to the mid-frontal plane of the structure including non-linear interface elements is used to study the influences of different assumptions for the interface. Both fully bonded and frictionless assumptions predict unrealistic interface behavior. In order to accurately study the stability of the uncemented implant in the host bone, the interface condition should be modeled with friction behavior. When fully bonded or frictionless assumption is used, the results of the model should be interpreted with care.
Chapter 5

Implant Stability Study Using an Anatomically Realistic Non-linear Bone-Implant Model

5.1 Introduction

In uncemented total hip arthroplasty, initial stability (in terms of relative motion between bone and implant) of the femoral component in the host bone is the most important factor to assure the success of the operation. Thigh pain occurs more frequently with uncemented than cemented implants, and that this is most pronounced when the femur is loaded in positions approaching horizontal such as standing up from a seated position or stair climbing [46]. At these positions, due to the gravitational force of the body weight pushes downward on the offset prosthetic head, a large torsional load is generated about the long axis of the stem and is not seen during level walking or single-leg stance during which axial loads dominate. The torsional load of this kind has emerged as the most important loading case to cause bone-implant interface debonding. While most studies in the literature evaluate stress distribution and relative motion with axial loading, a few experimental methods [57, 12, 47, 13] have also evaluated the problem with torsional loads. Investigations of stress distribution and relative motion at the bone-implant interface using finite element method were rarely reported in the literature. A better understanding of the interface behavior under torsional loads will benefit new design of prosthetic components with better stability in the host bone.

The purpose of this chapter is to study the stability of an implant under single-leg stance loads and torsional loads generated during stair climbing. The method developed in chapter 3 is used to construct an anatomically realistic bone-implant
model. The interface element INTER9 used in chapter 4 is included in the model to simulate the non-linear behavior of the interface. The loading condition for stair climbing will also be analyzed.

5.2 Literature Review

Crowninshield et al. [18] did a biomechanical investigation of the human hip during level walking, climbing and descending stairs, and rising from a chair. They found that in the cases of stair climbing and rising from a chair, the peak forces in the direction of progression were roughly two to three times the corresponding forces during level walking. They suggested that this force affect the amount of torque which is transmitted to the bone-cement interface of a hip prosthesis and may be related to prosthesis loosening.

Andriacchi et al. [5] experimentally analyzed the motions, forces and moments at the major joints of the lower limbs of ten men ascending and descending stairs. They found that the mean maximum net flexion-extension moments at the hip were 123.9 N-m during ascending stairs for a mean body weight of 71 Kg. The magnitudes of these moments are considerably higher than those produced during level walking. They suggested the forces and moments generated during stair climbing should be considered when establishing design criteria for prosthetic devices.

Phillips and Messieh [46] analyzed the clinical results of an uncemented hip replacement using a Moore stem on 41 patients. Most patients felt pain during the first few steps after rising from a sitting position and also worse when climbing stairs. Stem loosening was always most clearly demonstrated by rotational load.

Hodge et al. [27] measured pressure at the hip joint using an instrumented Moore-type prosthesis. The highest pressure was recorded while the patient was rising from a chair and it occurred in the superior and posterior aspects of the acetabulum.
Sugiyama et al. [60] experimentally investigated the role played by torsional loads in loosening of uncemented femoral components and in cemented components prepared by three different cementing techniques. They found that cemented components had better stability than uncemented component. Even the best cement technique is prone to failure in torsion when exposed to normal daily use.

Schneider et al. [57] developed an in vitro dynamic method for measurement of the relative motion of the femoral component of hip prostheses. Both axial and torsional loads were applied in their model. They confirmed that the use of cement reduces sinkage and rotation manyfold. Certain cementless implants may achieve stability comparable to cemented ones in some load directions.

Davy et al. [20] and Kotzar et al. [38] used an instrumented hip prosthesis to measure joint reaction force during daily activities. Their instrumentation can measure joint reaction force components in three orthogonal directions. This gave the magnitude, direction and location of the resultant force on the femoral head. They found the peak joint contact force during single-leg stance is 2.1 times body weight (BW); during rising from a chair it’s 1.23 BW; and during stair climbing the peak force is 2.6 BW.

Phillips et al. [47] did a series of experiments on femoral stems by putting femora horizontally and loading vertically (pointing posteriorly) to study the failure criteria and loosening load on the stems. They found the mean loosening torque was 23 N-m. In all specimens, failure began with an internal rotation of the stem within the femur and was followed by a combination of both internal rotation and posterior subsidence. Muscle forces were neglected in their experiments.

Burke et al. [12] experimentally evaluated the initial stability of cemented and uncemented femoral components during simulated single-leg stance and stair climbing. They found that both types were very stable in simulated single-leg stance. However,
in simulated stair climbing, the cemented components were much more stable than the uncemented components.

5.3 Methods

The same femur used in chapter 3 is used in this chapter with a Biomet [8], Mallory head prosthesis implanted. There were a total of 55 CT scans along the long axis of the femur. The length of the prosthesis is 193 mm. The last CT scan is only 15 mm distal to the tip of the stem. This slice is duplicated and translated 75 mm distally to extend the total length of cortical bone beyond the tip of the stem to 90 mm which is about 3 times the diameter of the distal bone cross-section. To construct the FE model, the general procedure shown in Figure 3.2 is followed.

5.3.1 Geometric Model

Only 15 slices were used to construct the geometric model. The cubic parametric lines generated from selected characteristic grids on the bone and implant contours for the geometric model are shown in Figure 5.1. From these lines, parametric patches and hyperpatches are generated. The geometric model composed of a set of hyperpatches is displayed in Figure 5.2. There were a total of 71 hyperpatches generated for the model. The mid-frontal plane of the model is at an angle of 17.59 degrees with the zz plane of the CT scan coordinates.

5.3.2 Finite Element Mesh

The major element type used in this model were 20-node quadratic hexahedral element. At the proximal transition region where cortical bone becomes thin, 15-node wedge elements were used and 8-node shell elements were used for the thin layer of cortical bone in the proximal region. There were 168 solid elements for the solid
cortical bone; 264 elements for the trabecular bone; 12 shell element for the thin layer of cortical bone; 216 solid elements for the implant; and 124 non-linear interface elements. The individual element groups are displayed in Figure 5.3. There are a total of 4013 nodes for the model with interface elements and 3155 nodes for the fully bonded interface model. From the experience with the idealized symmetric models in chapter 4 and Appendix A, finer mesh is used in the proximal region and near the tip of the implant.

5.3.3 Loads, Boundary Conditions, and Material Properties

Two loading conditions are studied in this chapter. The first case is simulating single-leg stance as before. The second case is simulating stair climbing. The loading condition simulating single-leg stance has been discussed in section 3.3.2.

The origin of the global coordinates is located at the CT scan origin of the most distal slice. The portion extended beyond the last CT scan has a negative z coordinate. Since the mid-frontal plane of the model is placed at an angle with the global xz plane, the force components for the joint contact force on the femoral head and abductor muscle force are transformed to the global coordinates of the model. The body weight (BW) of the patient are still assumed to be 71 Kg as before. Under single-leg stance loads, the components for joint contact force are -399.4, 126.6, and -1407 N in x, y, and z, respectively. The components for abductor muscle force are 398.5, -126.3 and 1148 N in x, y, and z, respectively.

The loading condition simulating stair climbing will be analyzed in the following. Since the muscle groups and their individual direction and force magnitude are not known during stair climbing, an idealized resultant muscle forces are assumed to act on the femur [43]. The free body diagram of the femur during stair climbing is plotted in Figure 5.4. Since the largest joint moment generated during stair climbing is in the
Figure 5.1  Parametric lines constructed from selected grids.
Figure 5.2  Hyperpatches for the geometric model.
Figure 5.3 Element groups for the FE model.
sagittal plane and moments on frontal and transverse planes are comparatively much smaller, only a 2D analysis in the sagittal plane is shown. The segmental weight ($W$) has a known magnitude and direction (see Figure 5.4(a)). The joint reaction forces at the hip and knee joint are decomposed into $x$ and $y$ components ($JX$ and $JY$). The resultant muscle force acting on the femur are hip flexors ($F(1)$) to flex the hip joint and knee extensors ($F(2)$) to extend the knee joint. These muscle forces can be translated to the joints with an equivalent force and couple (Figure 5.4(b)). After being translated to the joint, the resultant muscle force can be decomposed into $x$ and $y$ components and combined with the joint force. The sum of these two forces is the joint contact force ($RX$, $RY$). In a 3D analysis, the joint contact force and the joint moment all have three components. The magnitude and direction of the joint contact force and moment can be obtained from published experimental data as seen in the literature review section. These data are three-dimensional.

From Andriacchi et al. [5], the peak flexion moment generated by a 71 Kg man during stair climbing is 123.9 N-m in the sagittal plane and the adduction moment in the frontal plane is 25 N-m. There is no moment in the transverse plane. From Kotzar et al. [38], the peak joint contact force during stair climbing is 2.6 BW. After transforming to the global coordinates of the model used in this chapter, the force components are -494.5, 787, -1553.2 N in $x$, $y$, and $z$ respectively. The joint moment components after proper transformation are -110.545, 61.27, and 0 N-m in $x$, $y$, and $z$ respectively. The joint contact force component directing in the posterior direction is 601 N. This component generates a 21 N-m torque in the transverse plane about the long axis of the implant. This value is smaller than the 23 N-m failure torque measured by Phillips et al. [47].

The boundary condition for the model is the same as before. All nodes on the most distal cross-section are constrained in all directions. The Young’s moduli for
Figure 5.4 2D Free body diagram of femur during stair climbing.
cortical bone, trabecular bone, and metal implant are 14 GPa, .5 GPa, and 110 GPa. The Poisson's ratio is assumed to be 0.3 for all materials.

Two kinds of prosthetic surface finishes are simulated in this study. The first case is a smooth surface implant. The second case is a normal porous-coated implant. From Shirazi-Adl et al. [58], the friction coefficients for the smooth and porous-coated surfaces in contact with bone are 0.42 and 0.61 respectively. The fully bonded case is also studied for comparison purposes. Stresses predicted by the beam model proposed in chapter 2 are compared with the model with fully bonded interface.

5.4 Results

Model Verification

The convergence of the model is checked by the error estimator described in chapter 3. The fringe plot of the error for the model under single-leg stance loads is shown in Figure 5.5. The average error for the whole model is less than 1%.

Regions of Contact

The contact conditions for the interface elements are defined in the same way as in chapter 4. The regions of contact for the two loading cases and the two surface finishes are displayed in Figure 5.6. The upper row is the anterior view of the interface and the lower row is the posterior view. The regions of contact for the two surface finishes are almost the same under the same loading condition. But for the same surface finish, there are significant differences under the two different loading conditions. The contact regions under stair climbing loads are much smaller than those under single-leg stance loads. There are large regions of separation in the anterior-proximal section of the interface. The lateral side has larger contact regions than the medial side. The medial side has contact regions only in the most proximal region.
Figure 5.5  Fringe plot of the estimated error on the bone-implant model with fully bonded interface under single-leg stance loads.
The contact regions under single-leg stance show the similar pattern as in the symmetric model in chapter 4. There are regions of separation near the curve and straight stem junction and at the medial stem tip.

Relative Motions

The relative motions between bone and implant are calculated at four locations on the medial side of the interface. These sample nodes are illustrated in Figure 5.7. The first node is at the stem tip. The second node is in the middle of the straight section of the stem. The third node is at the curve and straight stem junction. The fourth node is at the proximal top of the interface.

The axial, transverse, and total relative motions at the four sample nodes under two loading cases are shown in Figure 5.8, 5.9, and 5.10, respectively. The relative motions for the smooth stem are generally larger than those for the porous-coated stem. The axial relative motions under single-leg stance loads are higher than those under stair climbing loads while the transverse component are just the opposite. Under single-leg stance loads, the axial components are generally larger than the transverse components while under the stair climbing loads the situation reverses. The maximum axial component is 60 microns occurs at the tip of the smooth stem under single-leg stance loads. The maximum transverse component is 165 microns occurs at the most proximal node (sample node 4) of the smooth stem under stair climbing loads. On average, the transverse relative motions under stair climbing loads are the largest among all cases.

Interface Stresses

The von Mises stresses in the trabecular bone under single-leg stance loads are shown in Figure 5.11. Those stresses under stair climbing loads are shown in Figure 5.12.
Figure 5.6 Regions of Contact in the interface under two loading cases.
Figure 5.7 Locations of sample nodes for relative motion calculations.
Figure 5.8 Axial component of the relative motion under two loading cases.
Figure 5.9 Transverse component of the relative motion under two loading cases.
Figure 5.10  Total relative motion under two loading cases.
The stress distribution patterns are similar for the two implant surface finishes. The stresses under stair climbing loads are higher than those under single-leg stance loads.

5.5 Discussions

The complex geometry of the bone-implant structure is difficult to model. With the aid of CT scan data, the geometry of the bone and implant is accurately defined. The procedure developed in chapter 3 is effectively applied here to construct the FE bone-implant model. The small error estimated by the error estimator verifies the adequacy of the FE mesh being used.

The prosthesis used in this model has no collar. The use of a collar can decrease axial motion but may not affect torsional interface motion (the transverse component) [63, 64]. The body weight (BW) of the patient is assumed to be 71 Kg, the same as the subject's BW in Andriacchi's paper [5] so that their data on joint moment can be directly applied. Joint contact forces data from Kotzar's paper [38] are precisely scaled according to the BW and transformed into global directions of the FE model. The advantage using a resultant muscle force approach in analyzing the loading condition simulating stair climbing is that individual muscle group and its force direction and magnitude need not to be known. Many models in the literature simulating stair climbing load do not consider muscle effect.

The similar pattern of regions of contact under the two different loading cases indicates that the regions of contact is not sensitive to the values of friction coefficient.

The total relative motions in the interface under single-leg stance loads are in the threshold range for relative motion (28 to 150 microns) suggested by Pilliar [48] that is needed to produce ingrowth of bone into porous-coated surface.

The large regions of separation in the interface under stair climbing loads verifies the observations by other experimental methods that this loading case is more
Figure 5.11 Von Mises stresses in the trabacual bone under single-leg stance loads.
Figure 5.12  Von Mises stresses in the trabecular bone under stair climbing loads.
threatening to the stability of the implant. This is further demonstrated in the relative motion study. The average high relative motion under stair climbing loads, especially the transverse component, is more likely to prevent bony tissues from ingrowing into implant surface. The maximum total relative motion (168 microns) occurs at the most proximal node under the stair climbing loads is over the maximum value (150 microns) suggested for bony ingrowth. This finding corroborates the findings of other authors using experimental methods that high torsional loads (still in the physiological range) should be applied to demonstrate differences in motion at the bone-implant interface and to produce motion that may be clinically important [12, 13]. This also suggest that the stability of a device can be evaluated best by torsional loading. In addition, the findings suggest that torsional loading of uncemented porous-coated devices should be avoided in early post-operative period (for 6 to 12 weeks, depending on the time needed for bony ingrowth) because relative motion that occurs at the bone-implant interface under high physiological loads such as during stair climbing can produce relative motion of the magnitude that inhibits ingrowth of bone. If prevention of relative motion at the bone-implant interface continues to be an important consideration in the design of uncemented devices, it may be beneficial to test the motion at the interface in the developmental stage for all uncemented femoral implants.

5.6 Comparison with Beam Model Results

The bone-implant model with fully bonded interface is also analyzed by the proposed beam model. There are a total of 55 CT scans along the entire length of the long axis. Every slice is used as a node along the beam model. The most distal slice is extended 75 mm distally as in the 3D solid model. In the region with slices containing both bone and implant, beam elements for bone and implant are generated separately. But
same node number is used for both materials so that they are simulated as having a rigid bond between them. Thus, the whole structure of the bone-implant system is modeled by the beam model as two separate beams bonded together as a composite structure. There are 56 elements generated for the bone and 51 elements for the implant and a total of 63 nodes (Figure 5.13).

The same loading conditions simulating single-leg stance and stair climbing are applied to the beam model. Same material properties for the materials are used in the beam model as in the 3D model. The beam model takes considerable less time to solve than the 3D solid model. For each loading case, it only takes less than 5 seconds to solve while a 3D solid model takes approximately 30 minutes. Octree models used in similar studies can take days of CPU time.

The von Mises stresses on the medial and lateral cortex of the cortical bone for the two models under single-leg stance loads and stair climbing loads are shown in Figure 5.14 and 5.15, respectively. Only the stresses up to the slices where cortical bone are still closed loops are shown. The results from the two model show very similar trends in stress profiles under both loading conditions. The difference near the most distal nodes is due to differences in defining the boundary conditions in the two models as discussed in section 3.5.3.

There are an average of 15 % of difference between the two models under single-leg stance loads in the region distal to $z = 30$ mm. Under stair-climbing loads, the differences in the von Mises stresses are larger quantitatively than under the single-leg stance loads. Overall, the beam model is in good agreement with the 3D solid model qualitatively in predicting stress profiles in the bone-implant system.
Figure 5.13  Beam elements and cross-sectional contour lines for the anatomic bone-implant model.
Figure 5.14 Comparison of von Mises stresses between beam model and 3D solid model under single-leg stance loads.
Figure 5.15  Comparison of von Mises stresses between beam model and 3D solid model under stair climbing loads.
5.7 Summary and Conclusion

An anatomically realistic uncemented bone-implant model is constructed in this chapter to study the stability of the implant in the host bone under single-leg stance and stair climbing loads. Smooth and porous-coated implant are modeled in this study. Non-linear interface elements are included to simulate the non-linear behavior of the bone-implant interface.

The regions of contact in the interface is insensitive to the friction coefficients assigned to the interface. The stair climbing loads generate torque around the long axis of the implant and large transverse relative motion which is very likely to inhibit bony ingrowth into the porous-coating of the implant. They have greater threat to the stability of the implant than the single-leg stance loads. The stair climbing loads should be applied to all uncemented femoral devices in the developmental stage to test their stability in the host bone. Torsional stability should be seriously considered and pursued when designing uncemented implants.

Stress profiles in the bone-implant system predicted by the beam model are in good agreement with the 3D solid model qualitatively.
Chapter 6

Conclusions

6.1 Summary

A 3D unsymmetric beam model is developed in this research. The model is unique in using variable cross-sectional properties calculated from CT scan, in coupling of the bending behavior in two orthogonal directions of unsymmetric bending, and in calculating torsional effect based on the torsional theory of thin-walled closed sections. The beam model is economical in total data preparation time and in CPU time to solve the model comparing to 3D solid model. The beam model is efficient in predicting the forces, deflection, moment, twist, stress profiles in the femur with or without an implant. It can be used as a preliminary design tool for quick parametric study of the bone-implant system.

A systematic method, using CT scan data and a computer-aided design tool to construct detailed, anatomically realistic 3D solid models of intact bone and bone-implant system, is also developed. Results from models constructed by this method are compared with experimental data and checked by an error estimator for their accuracy. It has been show that models constructed by this method are reliable and accurate for detailed 3D stress analysis of the bone-implant system.

The influence of different bone-implant interface assumptions on implant stability and stress distribution in bone are studied using idealized symmetric bone-implant model. Non-linear interface elements are included in the model to simulate the interface behavior. Fully bonded and frictionless assumptions for the interface are not realistic especially in the early post-operative period. Friction in the bone-implant interface should be modeled to more accurately approximate the real situation.
Influence of two loading conditions, single-leg stance and stair climbing on the stability of the implant are studied using a detailed 3D anatomic model. Joint reaction force and moment including muscle effects are considered. It is found that stair climbing loads are more threatening than the single-leg stance load to the stability of the stem, especially to the torsional stability. Stair climbing loads should be applied to new implant design to check its stability in the host bone.

Finite element modeling of the bone-implant system provides a useful method to study many complex factors involved in bone-implant system. From these models, new implant design features can be studied and the performance of the implant can be predicted before implantation.

6.2 Concluding Remarks

In this research, the material properties for bone are assumed to be isotropic. This might have some influence on the stress distribution in bone. Isotropy is true for metallic implant materials. Cortical bone, by reasonable approximation, can be considered linearly elastic and transversely isotropic. The elastic relationship for cortical bone can also be simplified from transverse isotropy to isotropy if the stresses and strains in the transverse and tangential directions are of lesser importance for the problem investigated [29]. The material properties for trabecular bone is anisotropic. To the first-order approximation, its elastic moduli can be expressed as a function of its porosity measured by its apparent density [14]. The elastic properties of trabecular bone also depend on the directionality of its structure. To more accurately model the bone properties, CT scan data can be used to compute site specific material properties for each element based on the relationship between bone density and Young’s modulus [21, 33].
The proposed beam model provides a useful preliminary design tool for parametric study of the bone-implant system. Since rigid bonding is assumed between bone and implant, its application is limited to prediction of stresses in models with this kind of bonding. The discrepancy between beam model and 3D solid model indicates that a 3D solid model is needed for more accurate stress analysis especially in the region proximal to the lesser trochanter.

Accurate approximation of the complex geometry of the bone and implant is the basic step toward success of the modeling. As observed in several analysis [31, 13], geometry, especially the implant shapes, plays an important role in stress distribution and load transfer in the bone-implant system. With the help of CT scan and PATRAN, every effort has been made to more closely model the original shape of the objects. The bottleneck for automating the model building process is to select proper characteristic grids from the bone and implant contours to accurately model their shapes. This depends on one's judgement in picking the proper grids. If the grid selection process can be done by a computer algorithm, the model building process has a great potential to be fully automated from geometric model construction to finite element mesh generation.

Model verification is always important for finite element models. Experimental data provide an excellent reference for FE model verification. FE models should always compare with experimental data whenever possible to verify their adequacy and accuracy. Many experimental methods provide information on only one facet of the behavior of the implant [57] as in some failure tests of the devices. FE models have advantages over experimental methods in their ability to do parametric studies and in the ease of applying complex loading conditions and boundary conditions.

Stability study of un cemented devices provides useful information for assessment of the adequacy of the implant design and prediction of their performance in the host
bone. These information are gaining more attention by surgeons and designers and published literature have been increasing in recent years. Most of these studies are done by experimental methods. The difficulties in FE modeling of the non-linear interface problems are the complexity in creating the properties of such a model and convergence problems with the non-linear interface elements.

The desire to produce prostheses with improved clinical performance continues to offer major design and analysis challenges to bioengineers. Finite element models will continue to play an important role in providing useful information for the design and analysis of these devices. An exciting new possibility is the incorporation of bone remodeling theories in FE models for bone-implant structures to analyze stress shielding effect. Another interesting possibility is using FE models to do numerical shape optimization of the implant. These studies will offer more insight to the design of implants with better performance.
Appendix A

Verification of the Non-linear Interface Element

The non-linear interface element used in this research is the element type INTER9 in ABAQUS. In order to test the accuracy of this element in predicting normal and shear stresses in the bone-implant interface, a test is carried out for the verification. The test model is adopted from [33]. The cross-sectional geometry, material properties, and loads used in that model were based on a model in the literature [53] which was a representative of a hip stem in the mid diaphysis. It offered an opportunity to verify the interface element used in this study.

The dimension, material properties, loading condition, and boundary condition of the model is illustrated in Figure A.1. The lengths of the bone and stem were 140 mm and 70 mm, respectively. The inner and outer diameters of the bone were 20 mm and 30 mm respectively, the outer diameter of the stem was 17 mm.

The element discretization, biased towards the stem tip, is the same as their model. The symmetry of the problem was exploited, so only half of the fully circular cross-section was modeled. The resulting model contained 280 solid, quadratic, isoparametric elements, 40 interface elements, and 1870 nodes (Figure A.2). A perfect fit is assumed between bone and implant. The interface elements have no thickness. The corresponding nodes on opposite faces of the interface element have same coordinates but different node number before loading.

Fixity boundary conditions enforce the kinematics of composite beam theory at the most proximal cross-section of the model. Therefore, the displacements perpendicular to this fixed plane were zero and shear stresses could not develop in the plane of this cross-section. Appropriate displacement boundary conditions were also ap-
Figure A.2  Finite element mesh for the axisymmetric model.
plied to model the symmetric behavior of the structure about a plane which bisected the cylinder in a direction parallel to the stem axis (Figure A.2).

A pure bending moment of 40 kN-mm was applied to the finite element model by applying equal forces in opposite directions to two nodes on the outer diameter of the bone at the most distal cross-section (Figure A.2). There were two region of homogeneous material properties (Figure A.1): the cortical bone (18 GPa) and the stem (200 GPa). The bone and stem are assumed to be linearly elastic and isotropic; a Poisson’s ratio of 0.3 was assigned for each material.

Three interface conditions were tested: fully bonded, no-tension no-friction, and no-tension with friction coefficient $\mu = 0.47$. The normal stresses and shear stresses in the stem across the interface are shown in Figure A.3 (medial side) and Figure A.4 (lateral side). These stress values duplicate those from their model. The validity and the accuracy of the interface element INTRER9 used in this research are thus verified.
Figure A.3  Normal stresses in the stem.
Figure A.4 Shear stresses in the stem.
Bibliography


