Subchondral Bone Properties of Femoroacetabular Impingement Patients

by

Ahmed Alnabelseya

B.Eng. in Mechanical Engineering, Carleton University, 2009

A thesis submitted to the Faculty of Graduate and Postdoctoral Affairs
in partial fulfillment of the requirements for the degree of

**Masters of Applied Science
in
Mechanical Engineering**

Carleton University
Ottawa, Ontario

© 2012, Ahmed Alnabelseya

Library and Archives
Canada

Published Heritage
Branch

395 Wellington Street
Ottawa ON K1A 0N4
Canada

Bibliothèque et
Archives Canada

Direction du
Patrimoine de l'édition

395, rue Wellington
Ottawa ON K1A 0N4
Canada

Your file Votre référence

ISBN: 978-0-494-91580-6

Our file Notre référence

ISBN: 978-0-494-91580-6

NOTICE:

The author has granted a non-exclusive license allowing Library and Archives Canada to reproduce, publish, archive, preserve, conserve, communicate to the public by telecommunication or on the Internet, loan, distribute and sell theses worldwide, for commercial or non-commercial purposes, in microform, paper, electronic and/or any other formats.

The author retains copyright ownership and moral rights in this thesis. Neither the thesis nor substantial extracts from it may be printed or otherwise reproduced without the author's permission.

AVIS:

L'auteur a accordé une licence non exclusive permettant à la Bibliothèque et Archives Canada de reproduire, publier, archiver, sauvegarder, conserver, transmettre au public par télécommunication ou par l'Internet, prêter, distribuer et vendre des thèses partout dans le monde, à des fins commerciales ou autres, sur support microforme, papier, électronique et/ou autres formats.

L'auteur conserve la propriété du droit d'auteur et des droits moraux qui protège cette thèse. Ni la thèse ni des extraits substantiels de celle-ci ne doivent être imprimés ou autrement reproduits sans son autorisation.

In compliance with the Canadian Privacy Act some supporting forms may have been removed from this thesis.

While these forms may be included in the document page count, their removal does not represent any loss of content from the thesis.

Conformément à la loi canadienne sur la protection de la vie privée, quelques formulaires secondaires ont été enlevés de cette thèse.

Bien que ces formulaires aient inclus dans la pagination, il n'y aura aucun contenu manquant.

Canada

Abstract

Osteoarthritis (OA) affects a large portion of the population and is considered to be the most common form of arthritis. It has been speculated that OA is a result of femoroacetabular impingement (FAI), a condition in which abnormal anatomy in the hip causes impingement of the femoral neck with the acetabulum. This impingement may lead to an increased subchondral bone stiffness in the femoral head and the acetabulum compared with healthy patients, which is thought to result in articular cartilage degeneration and, ultimately, OA. To verify this speculation, the effective isotropic elastic tissue moduli (E_{eff}) of six subchondral bone biopsies from cam-type FAI patients were determined and compared to that of a healthy control specimen in a combined experimental, and finite element analysis. Initially, models of the specimens were created using micro-computed tomography (μCT) techniques, and then meshed using 4-node tetrahedral elements. The bone specimens were then tested using three point bending to determine the force-displacement relationship of the bone structure. This data was used in a finite element analysis (FEA) simulating the three point bending test to obtain the E_{eff} value of each specimen. The E_{eff} of the FAI specimens varied between 1.2 GPa and 2.3 GPa whereas that of the control specimen was 1.0 GPa. The increased E_{eff} observed in the FAI specimens may have been due to an increased mineralization in the bone tissue and might have contributed to articular cartilage degeneration of the specimens.

Acknowledgements

First and foremost I would like to express my gratitude to Prof. Hanspeter Frei who offered me the opportunity to work with him and has helped guide me through this journey. He has been very inspirational and without his support and guidance this study would not have been successful. His methods have been nothing but helpful and his influence will forever have an affect on my career.

Furthermore, I would like to thank Andrew Speirs and Alejandro Guzman for sharing their invaluable knowledge and experience. This group has made my time here enjoyable and beneficial. I would finally like to thank the Mechanical and Aerospace Engineering department for providing the facilities and finances for this research project.

Many thanks to my beloved family and friends who have been supportive and understanding throughout the duration of my studies. I would like to dedicate this thesis to my beloved late grandmother, Yussra Moafi (1940-2012).

Chapter 1	12
Introduction and Background	12
1.1 Research Objectives	13
1.2 Biomechanics of the Hip	14
1.2.1 The Hip Joint	17
1.3 Osteoarthritis and Femoroacetabular Impingement	19
1.3.1 Femoroacetabular Impingement	20
<i>1.3.1.1 Types of FAI</i>	<i>21</i>
1.4 Bone	23
1.4.1 Composition of Bone Tissue	23
1.4.2 Bone Structure	24
1.4.3 Modeling of Cancellous Bone	27
<i>1.4.3.1 The Hierarchal Levels of Cancellous Bone</i>	<i>28</i>
<i>1.4.3.2 The Elastic Modulus of Cancellous Bone</i>	<i>29</i>
1.5 Literature Review	32
1.5.1 Experimental Analysis of Cancellous Bone	32
1.5.2 Computational Modeling of Bone	34
Chapter 2	36
Methodology	36

2.1 Overview of Methodology	36
2.2 Preparation of Bone Specimens	38
2.3 Model reconstruction	42
2.3.1 MicroCT Scanning	42
2.3.2 Model Segmentation	43
2.3.3 Volume Mesh Generation	44
2.3.3.1 Volume Reconstruction	44
2.3.3.2 Determining Specimen Porosity Values	45
2.3.3.3 The iso2mesh Algorithm	47
2.3.3.4 Meshing Using the iso2mesh Algorithm	49
2.4 Experimental testing	50
2.4.1 Setup and Calibration	51
2.4.1.1 Calibration of the CDMD	52
2.4.1.2 Calibration of the load cell	53
2.4.2 Experimental test	53
2.5 Finite Element Analysis	56
2.5.1 Input File	56
2.5.2 FEA preprocessing	57
2.5.2.1 Parts and Assembly	57
2.5.2.2 Material Properties	58

2.5.2.3 Boundary Conditions, Interactions and Applied Deflections	58
2.5.3 FEA analysis (post-processing)	61
2.6 FEM VALIDATION	62
2.6.1 Volume Mesh Validation	62
2.6.2 Threshold level validation	63
2.6.3 Element type Validation	65
2.6.4 Element Size Validation	66
2.6.5 Rigid Cylinder Validation	70
2.7 Specimen Models	70
Chapter 3	73
Results	73
3.1 Porosity of the Specimens	73
3.2 Experimental Results	74
3.3 Finite Element Analysis Results	78
Chapter 4	80
Discussion	80
4.1 The Relationship Between Osteoarthritis and FAI	80
4.2 Experimental and FEA Results	81
4.3 Error Analysis	82

4.4 Limitations	84
4.5 Future Work	85
Chapter 5	86
Conclusion	86
5.1 Conclusions	86
5.2 Contributions	87
References	88
Appendix A: Mesh Validation: CT and Mesh Alignment	100
Appendix B: MTS Step Definitions	103
Appendix C: Calibration Data and Results	104
Appendix D: Equations used to calculate Estructural	106
Appendix E: Equipment and Apparatus	108
Appendix F: Abaqus Time Step Increments for Applying Displacement in FEA	109
Appendix G: Matlab Algorithms	110
Mesh	110
Volume.m	111
mat2Inp	112
porosity	113

List of figures

- 1.1 Human skeletal system
- 1.2 The anatomical coordinate system
- 1.3 The human hip
- 1.4 Section into the femoral head-acetabulum interface
- 1.5 Different types of FAI
- 1.6 Impingement caused in FAI hips
- 1.7 Collagen fibers in bone
- 1.8 View of the inside of bone
- 1.9 Cross section of the human femur showing the trabecular structure within
- 1.10 Different hierarchal levels of cancellous bone
- 1.11 Stress-strain curves of cancellous bone specimens of varying densities
- 2.1 Flowchart showing approach taken in the analysis
- 2.2 Model of left femur illustrating the location from which the tissue samples were extracted
- 2.3 Bone specimens as extracted by patients, after being cut, and after removing cartilage layer
- 2.4 Slice number 945 of 1016 from the micro-CT scanning of sepcimen 5 as viewed in Osirix Imaging Software v3.8.1
- 2.5 Cross sectional slice 945 of 1016 of specimen 5 before and after thresholding
- 2.6 3D rendering of specimen 1

- 2.7 Cross sectional images of specimen 1 superimposed to show the bounding surface used in calculating the porosity
- 2.8 Topological errors that may arise during the meshing process
- 2.9 Illustration of surface mesh and corresponding volume mesh
- 2.10 Image of CDMD showing location of strain gauge
- 2.11 Three point bending test setup
- 2.12 Assembly used in finite element analysis
- 2.13 Boundary conditions applied to finite element model
- 2.14 Contact surfaces in finite element model
- 2.15 Specimen 1 before and after simulation
- 2.16 Micro-CT scans and meshed volume of specimen 1 aligned in Amira 5.3.3
- 2.17 Mesh models with varying threshold values
- 2.18 Varying degree of detail as a result of the number of elements in the mesh
- 2.19 Convergence test performed for finite element analysis of specimen 1
- 2.20 Meshed models of specimens 1 - 7
- 3.1 Force vs displacement graphs of specimens 1-7 obtained from three point bending of the specimens
- 3.2 Graph of porosity vs $E_{\text{structural}}$
- 3.3 Graph of porosity vs E_{eff}
- 4.1 Graph of force vs displacement using both the LVDT and CDMD to measure displacement
- A1: Specimen 1 Micro-CT scans and meshed volume
- A2: Specimen 2 Micro-CT scans and meshed volume

- A3: Specimen 3 Micro-CT scans and meshed volume
- A4: Specimen 4 Micro-CT scans and meshed volume
- A5: Specimen 5 Micro-CT scans and meshed volume
- A6: Specimen 6 Micro-CT scans and meshed volume
- A7: Specimen 7 Micro-CT scans and meshed volume
- B1: Step definition for experimental analysis
- C1: Pre-testing CDMD calibration graph
- C2: Post-testing CDMD calibration graph

List of Tables

- 2.1 Age and gender of patients from which biopsies were extracted
- 2.2 Dimensions and aspect ratios and apparent second moment of area of bone specimen
- 2.3 Calibration equations along with r^2 and squared error values for calibration lines
- 2.4 Simulation results for specimen 1 using different threshold ranges
- 2.5 Simulation results for specimen 1 using both 4-node and 10-node tetrahedral elements
- 2.6 Results from convergence test including % error value
- 2.7 Number of elements used to mesh each specimen volume model
- 3.1 Porosity values of the specimens
- 3.2 Total elastic displacement and corresponding reaction force of each specimen during three point bending
- 3.3 The effective isotropic tissue modulus of each of the seven specimens as obtained through the analysis
- C1: Voltage output from CDMD strain gauge as a result of displacement applied by micrometer (Before testing)
- C2: Voltage output from CDMD strain gauge as a result of displacement applied by micrometer (After testing)
- D1: Values obtained for Estructural calculations
- F1: Change in fraction of applied displacment with FEA step time

Chapter 1

Introduction and Background

Osteoarthritis (OA) is a debilitating and painful joint disease that involves damage to the cartilage layer covering the ends of bone in joints, due to wear and tear. OA affects a large portion of the population and is considered the most common form of arthritis [1]. It is estimated that 13% of the Canadian population (4.4 million) [2] and 12.1% of the U.S. population (37.7 million) [3] are affected by OA. With an aging population these numbers are likely to increase.

Although not fully understood, OA is directly related to cartilage degeneration. The literature suggests that cartilage degeneration in the hip joint is initiated by an increased stiffness of the layer of bone beneath the cartilage, known as subchondral bone [4]. Radin et. al proposed that excessive loading of the joint can cause microfractures within the subchondral bone [4,5]. As a result, the healing process proceeding bone microdamage is thought to stiffen the subchondral bone in that region. This in turn, can lead to increased stress in the articular cartilage, which over time can cause cartilage degeneration [6].

There is an increasing body of literature that suggests that OA may be caused by a condition known as femoroacetabular impingement (FAI) [7,8,9]. FAI is a condition where abnormal anatomy of the proximal femur results in an

impingement of the femoral neck with the acetabular rim and impedes on the regular range of motion of the hip. In FAI patients, this impinging contact can increase the stresses on the articular cartilage and cause cartilage degeneration [10]. Diagnosis of FAI would allow for early intervention, before major damage occurs, and could prove beneficial to both patients and the Canadian Healthcare system.

1.1 Research Objectives

The main objective of this thesis is to determine the mechanical properties of the subchondral bone of FAI patients with cam deformities. These properties are then compared with the subchondral bone of normal patients. This study is part of a multidisciplinary research effort to identify the natural history of FAI and to identify 'at-risk' patients who require corrective surgery. The data obtained from this research may help identify disease onset and progress and will be further used in patient specific FEA to identify 'at-risk' patients.

The following section will give a brief description of the human skeletal system, the hip joint, and continue to discuss OA and FAI.

1.2 Biomechanics of the Hip

The adult human skeleton consists of different bones that provide support and are joined together to form articulations and permit movement. The adult human skeletal system primarily consists of two types of tissue: bone and cartilage. While bone provides the human body with rigidity and structural support, cartilage, which is mainly found in joints, acts to shield the bone from direct contact. Cartilage provides a smooth surface at the joints and thus reduces friction between the bone during movement.

The human skeletal system is illustrated in Figure 1.1. In order to unambiguously describe location and orientation of bones, the 'anatomical coordinate system' is used. The three axes of the anatomical coordinate system are the anteroposterior axis, mediolateral axis, and the superior-inferior axis as shown in Figure 1.2. The different planes of the anatomical system are illustrated as well.

Figure 1.1: Illustration of the human skeletal system [11]

Figure 1.2: The anatomical coordinate system indicating the coordinate axes and planes [12]

1.2.1 The Hip Joint

The part of the human skeletal system that is of interest in this study is the hip joint. The hip joint is classified as a synovial joint, or diarthrosis (one in which the bones are not directly joined), allowing for a greater degree of mobility compared to other joints [12]. From a biomechanical point of view, the hip joint can be modeled as a ball and socket joint as it permits rotation in all three angular directions. The hip joint is illustrated in Figure 1.3.

In the hip joint, the femoral head articulates with the acetabulum of the pelvis. The femoral head constitutes the medial-superior (uppermost and innermost) part of the proximal femur. The narrow region of the femur that connects the femoral head to the rest of the femur is known as the femoral neck (Figure 1.3). The concave cavity in the pelvis in which the femoral head lies is known as the acetabulum.

Figure 1.3: The human hip showing its different parts [14]

The acetabulum is located on the anterior, inferior, and lateral sides of the pelvis. At the area of contact between the femoral head and the acetabulum, a layer of cartilage, known as articular cartilage, exists to reduce frictional effects within the joint. In the absence of articular cartilage on either of the contact surfaces, the coefficient of friction between the surfaces increases, resulting in pain and/or further damage during motion [15]. The articular cartilage on the femoral head and that on the acetabulum are illustrated in Figure 1.4. The cartilaginous rim surrounding the acetabulum is known as the acetabular labrum. Between the two layers of cartilage, a lubricating fluid, called synovial fluid, exists to further decrease friction in the joint. The bone beneath the articular cartilage is

known as subchondral bone and was shown to play a significant role in the articular cartilage degeneration process [16].

Figure 1.4: Section into the femoral head–acetabulum interface. The cartilage layer can be seen to separate between the bone in the joint [17]

1.3 Osteoarthritis and Femoroacetabular Impingement

Hip pain is a common problem which can have a number of underlying causes, the most common of which is osteoarthritis (OA). According to Osteoporosis Canada, OA is defined as “a degenerative joint disease that involves thinning or destruction of the smooth cartilage that covers the ends of bones, as well as changes to the bone underlying the joint cartilage” (i.e.

subchondral bone). Osteoarthritis results in pain, joint stiffness, and reduced range of motion of the affected joint, which ultimately affects one's ability to do physical activities and reduces overall quality of life [1].

1.3.1 Femoroacetabular Impingement

OA of the hip is characterized by cartilage degeneration, which ultimately results in direct bone contact between the femoral head and acetabulum. The development of cartilage degeneration is not fully understood; however, it is speculated that Femoroacetabular Impingement (FAI) is a possible cause. As defined by Ganz et. al, FAI is a condition where abnormal morphological features, present in either the proximal femur and/or the acetabulum, result in atypical contact, which can lead to excessive wear of the joints [8]. These abnormal features include excessive bone growth present in the femoral neck, the acetabulum, or both locations (Figure 1.5). As movement occurs between the femoral head and acetabulum, this excess bone rubs against the articular cartilage in the joint generating stress between the two contact surfaces (Figure 1.6). This contact may cause damage or wear to the cartilage possibly leading to its degeneration and the development of OA.

FAI can be symptomatic or asymptomatic. FAI is considered asymptomatic when no pain or change in the range of motion of the hip occurs despite the presence of abnormal deformities. Symptomatic patients suffering from FAI experience hip pain and restriction in the range of motion of the hip

joint. Treatment consists of physiotherapy and modifications to the patient's activities to avoid certain movements that cause impingement and pain. Surgical treatment consists of surgical correction of the deformity which is aimed to relieve some or all of the symptoms of FAI and even prevent further damage to the joint. In severe cases with the presence of advanced cartilage degeneration a total hip arthroplasty or hip resurfacing may be necessary [8,18-21]

1.3.1.1 Types of FAI

FAI has been classified into three types, depending on where the excess bone is located; cam, pincer, and a mix between the two (Figure 1.5). Cam type FAI is caused by contact of the femoral head into the acetabulum during motion due to the aspherical shape of the femoral head (Figure 1.6). This abnormal contact results in shear forces that produce abrasion of the acetabular labrum, which may eventually lead to tear or detachment of the labrum. Pincer type FAI is caused by contact between the acetabular rim and the femoral neck due to abnormal features present in the acetabulum. The main damage occurs in the acetabular labrum and further contact may cause degeneration of the labrum and adjacent articular cartilage [8]. In some cases, both cam and pincer type of FAI exist together.

Figure 1.5: Types of FAI [22]

Figure 1.6: Impingement caused by CAM type FAI [23]

1.4 Bone

From a macroscopic point of view, bone is a structure and has a number of functions in the human body including support and movement of the body, as well as the protection of vital soft organs. To carry out these functions, bone needs to be strong and stiff. Properties vary throughout the bone and depend on both its composition and structure.

1.4.1 Composition of Bone Tissue

Bone is a highly vascular, mineralized connective tissue that has a dry weight composition of about 25% organic materials (mainly collagen fibers), and 75% inorganic minerals, (mostly calcium and phosphate) [24]. Collagen comprises approximately 50% of the bone by volume and 25% by weight [25]. It has an internally cross-linked structure with transverse spacings between its fibrils (Figure 1.7). Due to its structure, collagen contributes greatly to the bone's elasticity and tensile strength and is where most of the strain energy is stored during loading. Mineral salts make up most of the remaining 50% of bone tissue volume (and 75% of the remaining weight). These salts, primarily calcium and phosphate, are located within the collagen spacings and are responsible for the hardness, rigidity, and strength of the bone [12].

Figure 1.7: Collagen fibers in bone [12]

1.4.2 Bone Structure

Bones are structures made of porous tissue with varying porosity levels. According to its porosity, the bone structure can be classified into two types; cortical (compact) bone, and cancellous (porous) bone. Most bones consist of an outer layer of dense *cortical* bone that encloses a core or porous *cancellous* bone structure. This combination gives bone its strength while maintaining a light weight. The outer cortical bone protects the inside structure from fracture due to its rigidity, while the inside cancellous bone provides a strong yet light framework, and is capable of absorbing relatively high impact loads [24].

The basic structural unit of cortical bone is the osteon, shown in Figure 1.8. Osteons are roughly 200 μm in diameter, and are usually aligned with the long axis of the bone [24]. An osteon possesses a cylindrical shape and is made of concentric layers called lamellae. At the center of each osteon is a channel (the haversian canal) through which blood vessels run [14]. Blood vessels extend within the osteon through radial canals, called Volkmann's canals, that run throughout the entire bone structure. These channels and canals allow a large amount of blood to enter and exit the bone. This high blood flow grants bone an excellent capacity for self repair and gives it the ability to alter its properties and configuration according to the loads under which it is placed [14].

Figure 1.8: A view of the inside of bone. Figure 1.8a indicates the locations of cortical and cancellous bone. [26]. Figure 1.8b shows the cortical bone clearly illustrating the Haversian canals within the osteons [27]

Cortical bone has a porosity ranging from about 5% to about 30% [28]. Osteons are tightly packed within the cortical structure as shown in Figure 1.8. It is this structure that gives cortical bone its anisotropic properties. Looking inwards from the outer layer of bone, the porosity of the bone structure increases and the dense packing of osteons gradually turns into a loose mesh structure (Figure 1.8a). This highly porous mesh is referred to as cancellous bone. Cancellous bone has a cellular structure and is composed of an interconnected series of thin plates and rods (trabeculae) with a porosity level ranging from about 30% to more than 90% [25,29]. Like cortical bone, the rods and plates of cancellous bone are made up of lamellae. Figure 1.9 shows the cancellous bone structure in the proximal femur. It is shown how the trabecular structure of cancellous bone is open and consists of rods and plates.

Figure 1.9: A cross section of the human femur showing the trabecular structure of cancellous bone contained within, and the surrounding denser cortical bone along the perimeter of, the proximal femur [27]. A magnification of the cancellous bone in the femoral head is shown as well [30]. The porosity of cancellous bone in the femoral head is reported to be 75% [31]

1.4.3 Modeling of Cancellous Bone

The focus of this research is the subchondral bone of the femoral head which is mostly made up of cancellous bone. Cancellous bone, as described above, has a cellular structure [29, 32]. Modeling cancellous bone can be done at various scales, with varying mechanical properties. Three hierarchical levels of trabecular bone have been defined in this study as illustrated below (Figure 1.10).

1.4.3.1 The Hierarchical Levels of Cancellous Bone

The *microscale* level (Figure 1.10a) considers a single lamella in the trabeculae of cancellous bone. Composed of mineralized collagen fibers, each lamella spans a thickness of approximately 3-7 micrometers. Grouped together, these lamellae form the single trabeculae of cancellous bone at a *tissue* level (Figure 1.10b). The lamellae are oriented in different directions and are held together at cement lines, which form around them. The average thickness of a newly formed trabecula is about 50 micrometers. Trabeculae can have rod or plate like shapes and, connected together at a *structural* level (Figure 1.10c), form the porous, trabecular structure of cancellous bone.

Figure 1.10: Different Hierarchical levels of cancellous bone. (a) At a microscale level, an individual lamella can be seen. (b) Grouped together, the lamellae form the bone tissue of cancellous bone at the tissue level. (c) At a structural level, the individual trabeculae come together as rods and plates to form the porous structure of cancellous bone [33]

The mechanical properties of cancellous bone would greatly differ depending on the level at which it is analyzed. At a *microscale* level, the properties considered would be those of the individual lamella unaffected by its connectivity to the others. At a *tissue* level, however, the cement lines between the lamellae have different properties and do affect the mechanical properties of the tissue. Similarly, the individual trabeculae have mechanical properties that differ from those of cancellous bone at a *structural* level.

1.4.3.2 The Elastic Modulus of Cancellous Bone

When defining the elastic properties of cancellous bone, it is important to recognize that these properties are quantified differently depending on the hierarchical level that is considered. At a tissue level, the tissue level modulus (E_{tissue}) is equivalent to the modulus of elasticity of the individual trabeculae. E_{tissue} is not affected by the geometry, and porosity of the trabecular structure, rather it is the composition of trabeculae that determines its value.

The structural modulus ($E_{\text{structural}}$) is the elastic modulus of the trabecular structure. $E_{\text{structural}}$ is usually determined by analyzing cancellous bone at a structural level and is greatly affected by the porosity and trabecular geometry as well as the tissue level properties of the individual trabeculae.

The stress-strain curve due to compressive loads, of cancellous bone has three distinct regions when loaded (Figure 1.11). Initially, the load profile is

linearly elastic and occurs as the cell walls of the bone structure axially bend. The slope of the linear region is considered the $E_{\text{structural}}$ of the bone specimen. Further loading causes the cells to collapse until the cell walls meet and touch, resulting in a roughly constant stress. Finally, after the cell walls meet, the resistance to load drastically increases, resulting in a final steep portion to the stress-strain curve. We hypothesize that the subchondral bone from FAI patients behave similarly .

Figure 1.11: Stress - Strain curves for cancellous bone in compression for varying densities. [32]

Bone, as mentioned earlier, is a two-phase “composite” material that is composed of collagen and mineral content. At a tissue level, the elastic

properties of bone are determined by these two components. The toughness of the tissue is due to the collagen in the bone, which is where most of the strain energy is stored. However, the strength of the bone tissue, and the slope of the linear (elastic) portion of the stress-strain curve, is mostly determined by the stiffening effect of the mineral content in the bone. Burstein et. al. showed that 97% of the elastic modulus of bone is determined by the bone's mineral content and that the collagen in the bone had little effect on the overall modulus [34].

When determining the elastic modulus of a two phase material, such as bone, the classical law of mixtures, developed by Voight, may be used [35]. The elastic modulus of a two phase material is directly related to the moduli and volumes of its individual phases. Equation 1.1 shows this relationship:

$$E_C = E_A V_A + E_B V_B \quad \text{Eq 1.1}$$

Where E_C is the modulus of elasticity of the two phase material

E_A is the modulus of elasticity of the first phase

E_B is the modulus of elasticity of the second phase

V_A , and V_B are the volume fractions of each phase.

Since the elastic modulus of the mineral content is much higher than that of collagen (approximately 38 times higher [34]) it can be assumed that any

increase in the bone's tissue modulus of elasticity would be due to an increase in the mineral volume.

In this thesis, a combined *tissue* level and *structural* level analysis was performed to obtain the effective isotropic tissue modulus (E_{eff}) of the subchondral bone of FAI patients. E_{eff} is defined as the isotropic and homogenous tissue level elastic modulus. Although the tissue modulus likely varies within the trabecula, this assumption has been made in similar analyses [36-38].

1.5 Literature Review

1.5.1 Experimental Analysis of Cancellous Bone

The mechanical properties of cancellous bone, namely the modulus of elasticity, have been determined through various types of mechanical tests, including three point bending, nano-indentation, compression, and microtensile tests. Combining experimental tests with computational finite element analysis (FEA) is a common approach taken and has been performed most often [39]. For healthy cancellous bone, values for the effective isotropic elastic modulus have been reported to be anywhere between 0.1 and 4.5 GPa [40]. Modulus values of cancellous bone from the femoral neck were reported to be below 1.4 GPa [41].

This value provided an early indication of the effective isotropic elastic modulus of healthy subchondral bone in this region.

Three point bending was chosen as the testing method in this analysis mainly due to the fact that it provided a more realistic loading profile compared to tensile or compression testing. In vivo loading of the femoral neck of FAI patients during motion occurs when the abnormal bump on the proximal femur comes in contact with the acetabular rim. This contact is better simulated with three point bending compared to pure compressive or tensile loading. In addition, the size of the available specimens are roughly 10 x 2.5 x 2.5 mm which makes pure compressive or tensile testing very challenging.

Based on specimen size, nano-indentation would have been a candidate method of testing. However, the microstructure of cancellous bone varies substantially [42] and the presence of cement lines between the lamellae of the individual trabeculae makes the measurements sensitive to location and a significant number of tests would be required to obtain an effective elastic modulus (E_{eff}) of the subchondral specimens.

Three point bending is a common method used to determine the mechanical behavior of bone, and has been used by Leppänen, Schriefer, and Gustafson, to characterize the mechanical properties of animal bone [43-45]. The same was done for human bone by Choi [46]. Obtaining the modulus of elasticity

of the bone specimens from the recorded load-displacement curves required either an analytical (usually beam theory) or numerical (usually FEA) solution method. Lenthe et al. compared bone tissue modulus of human femora, determined using beam theory to those using FEA. He found that by using beam theory, tissue modulus was underestimated for all femoral specimens tested [36]. The reason for this discrepancy is mainly due to the nonuniform cross-sectional area of cancellous bone that can only be accurately represented when creating a volumetric representation of the specimen.

1.5.2 Computational Modeling of Bone

A recently developed strategy to obtain the effective modulus of elasticity of bone tissue consists of using finite element models to simulate the load-displacement behavior from mechanical tests [47-49]. Ladd et. al combined three point bending with FEA and determined the elastic modulus of human L1 vertebrae (composed of cancellous bone) [50]. An important aspect of using FEA for bone tissue analysis is the accurate representation of the bone specimen in the finite element (FE) model, particularly with cancellous bone. The level of detail of the individual trabeculae in the model can have a significant impact on the simulation results [51]. For this reason, micro computed tomography (μ CT) techniques have become a common imaging method used in the creation of computerized bone models [52-54].

Today, μ CT scans are available with resolutions as low as 2 μm [52]. Ulrich et .al, however, reported no significant change in elastic modulus values obtained from models with resolutions below 28 μm [55]. Obtaining volume meshed models is usually done by converting groups of pixels into individual elements [56-58].

Chapter 2

Methodology

2.1 Overview of Methodology

The objective of this research was to determine the effective isotropic tissue modulus (E_{eff}) of subchondral bone in the femoral heads of patients suffering from Femoroacetabular Impingement (FAI). The approach taken to determine E_{eff} of each bone specimen is outlined in Figure 2.1. Subchondral bone specimens were extracted from tissue samples obtained from the femoral heads of FAI patients (Figure 2.2). Similarly, a control specimen was obtained from a cadaveric femur specimen with no signs of FAI. Micro-Computed Tomography (μ CT) scans of all the bone specimens were taken. These scans were then segmented and stacked to construct a finite element volume mesh with tetrahedral elements. To determine the properties of the specimens, a destructive three-point bending test was performed on each specimen. Finally, a finite element analysis, simulating the three-point bending test, was performed for each of the specimens using Abaqus CAE v6.10-2 [Simulia, Providence, RI]. The boundary conditions from the experimental three-point bending test were applied to the model and an iterative simulation process to match the experimental results was performed to obtain the modulus of elasticity of the corresponding specimen. The bone specimens, model reconstruction, experimental analysis, and finite element analysis are described in detail in Sections 2.2 to 2.5. A validation of the methodology was performed according to Section 2.6.

2.2 Preparation of Bone Specimens

Figure 2.2: a) FAI patient and b) healthy control model of left femur illustrating location of the femoral head from which the tissue samples were extracted

Subchondral bone specimens were obtained from six symptomatic FAI patients who underwent resection surgery of the cam-deformity (Figure 2.2a). An additional control specimen was extracted from a healthy cadaveric femur (Figure 2.2b), for a total of seven specimens. The bone specimens were extracted by Dr. Paul Beaulé from the Division of Orthopaedic Surgery at the University of Ottawa. Ethics approval was obtained from the Ottawa Hospital Research Board and Carleton University Research Ethics Board for this investigation. The age and gender of the hips from which the specimens were extracted are shown in Table 2.1. The bones were extracted using a cylindrical

rotating saw with an inner diameter of 1 cm. Since the specimens were obtained from patients, the size of each specimen was limited and larger specimens could not be obtained.

Table 2.1: Age and gender of each patient from which the biopsies were extracted

Specimen	Age (years)	Gender
1	25.5	Male
2	38.9	Male
3	29.5	Male
4	31.2	Male
5	46.9	Male
6	40.7	Male
7 (Control)	50	Male

Each specimen was composed of two layers; a layer of subchondral bone, and a layer of cartilage (Figure 2.3). The cartilage layer was used for a related study where the properties of cartilage were determined. Each specimen was cut into a beam shape, using a Buehler Isomer low speed diamond tip saw [SN 347-IS-8005, Buehler Tool Works Inc., Lake Bluff, IL], and the cartilage layer removed using a surgical scalpel before testing. The bone specimens were cut to obtain flat contact surfaces for the three-point bending test and to achieve the largest possible 'span length to thickness' aspect ratio [59].

For three-point bending 'span length to thickness' aspect ratio of 20:1 is recommended but this was not possible due to size limitations of the biopsies. A solid material with an aspect ratio of 5 undergoing bending was reported to have resulted in an underestimation of 29% in the measured elastic modulus due shear stresses, which are neglected in simple beam theory [44]. However, using FEA, shear stresses as any geometrical deviation from an ideal beam were accounted for, by an accurate model of the specimen and the three-point bending test.

During cutting the specimens were sprayed with saline to prevent friction induced heating. Figure 2.3 shows one of these bone specimens. The dimensions of each specimen after cutting were measured using a vernier caliper and are shown in Table 2.2, along with the aspect ratio and apparent second moment of area of the cross-section of the beam (I_x) of each specimen. The dimensions are reported as approximate values as the specimens were not perfectly beam-shaped. The same coordinated system defined for the FE model was used to determine I_x , where the z-axis is oriented along the length of the specimen.

Figure 2.3: Bone specimen (a) as extracted from patients, (b) after being cut, and (c) after removing cartilage layer

Table 2.2: Dimensions, aspect ratios, and apparent second moment of area (I_x) of beam shaped bone specimens

1	9.5	2.4	1.4	7.8	5.6	0.5
2	10.5	1.7	1.6	7.8	4.9	0.6
3	9.8	2.3	1.2	7.8	6.5	0.3
4	10.5	1.9	1.5	7.8	5.2	0.5
5	10.5	2.3	1.9	7.8	4.1	1.3
6	10.1	2.5	1.7	7.8	4.6	1.0
7 (Control)	10.4	2.8	1.6	7.8	4.9	1.0

2.3 Model reconstruction

A meshed volume model of each specimen was required to perform a finite element analysis. The steps taken to obtain each model were as follows:

1. Obtaining a μ CT scan series of each specimen
2. Segmenting the scanned 2D cross sectional images to distinguish the bone regions
3. Stacking the segmented 2D scan images to obtain a volumetric model of each specimen
4. Meshing the volumetric model

2.3.1 MicroCT Scanning

Scanning was performed using micro computed tomography techniques. The specimens were scanned using a SkyScan 1072 X-ray CT scanner [Skyscan, Kontich, Belgium]. The scans were taken at an in-plane resolution of 9 μ m and with an image spacing of 9 μ m. The results of each scan were 1016 16-bit grayscale (0 to 65535 gray scale values) cross sectional images of each specimen that were scaled down to 8-bit (0 to 256 gray scale value) images since 256 gray values gave a sufficient sensitivity to distinguish the bone tissue from the background. Figure 2.4 shows a sample cross sectional slice of one of the specimens.

Figure 2.4: 8-bit Slice number 945 of 1016 from the micro-CT scanning of specimen 5 as viewed in Osirix Imaging Software v3.8.1

2.3.2 Model Segmentation

“Image segmentation is defined as the partitioning of an image into non-overlapping, constituent regions that are homogeneous with respect to some characteristic such as intensity or texture” [60]. A segmentation process was required to clearly identify the boundaries of the bone specimens within the individual scans. Thresholding is a common segmentation process used in medical imaging that involves partitioning individual pixels in an image according to their corresponding intensity values. Thresholding was found to be a suitable segmentation technique [61], and was implemented using Osirix Imaging Software v3.8.1 imaging software (Pixmeo, Geneva, Switzerland). During the thresholding process, individual pixels in each slice were marked as either a bone pixel or a background pixel, based on CTgray level intensity. A threshold range with a grayscale value (GSV) between 60 and 80 was set to obtain a clear

bone to background contrast. A number of threshold range values were examined with negligible effect on the simulation results (see Section 2.6). Figures 2.5a and 2.5b illustrate how thresholding clearly distinguishes bone regions. From the figures, the surface definition of the non-thresholded slice (2.5a) is not as clear as that after thresholding is applied. This difference occurs because more noise pixels, due to Compton scattering of the X-rays, electromagnet effects from the scanner, X-ray energy, and other effects [62-64], are viewed in the non-thresholded image. This noise tends to distort the image, resulting in a rougher surface.

Figure 2.5: Cross sectional slice 945 of 1016 of specimen 5 (a) before and (b) after thresholding as viewed in Osirix Imaging Software v3.8.1

2.3.3 Volume Mesh Generation

2.3.3.1 Volume Reconstruction

To obtain three-dimensional models of the bone specimens, the segmented images from each μ CT scan series were stacked one on top of the

next. Using Matlab (Mathworks Inc., USA), an algorithm (volume.m) was developed to perform the model reconstruction. Algorithm 'volume.m' can be found in Appendix G. The algorithm was designed to read in a series of 2D matrices and "stack" them one after the next to create a volume rendering (3D matrix) resembling each of the specimens. Figure 2.6 shows one of the resulting models.

Figure 2.6: 3D rendering of specimen 1 obtained by stacking individual micro-CT slices using the 'volume.m' Matlab algorithm

2.3.3.2 Determining Specimen Porosity Values

The porosity of bone is defined as the ratio of pore volume to the total volume of the bone sample. The porosity value of each specimen was obtained by calculating the ratio of non-bone pixels in the 3D model to the total number of

pixels in that model. The CT slices were superimposed to identify the outline of the each model. A bounding volume was created just inside the image of the superimposed slices to ensure that all of the pixels considered in the porosity calculation were contained within the model (Figure 2.7).

Figure 2.7: Cross sectional images of Specimen 1 superimposed to show the bounding surface. The image shows all 1016 micro-CT slices from one of the psecimens superimposed. The area enclosed denotes that taken into consideration when calculating the porosity of that specimen.

2.3.3.3 The *iso2mesh* Algorithm

Fang et al. devised an algorithm (*iso2mesh*) that creates volumetric meshes from 3D binary or gray-scale images. The input parameters of the algorithm include the 3D image to be meshed, the *isovalues* parameter (the maximum number of voxels contained in each tetrahedral edge), and the *opt* parameter (the maximum number of voxels enclosed by each tetrahedral element) [58].

In the first step of the *iso2mesh* algorithm, a surface mesh is generated from a 3D matrix created by stacking a series of cross sectional images. A 'surface simplification' approach is used to extract the surface mesh where a voxel-resolution triangular surface mesh bounding the 3D image is created. A following mesh simplification process is then applied to re-mesh the surface based on the *isovalues* input parameter in what is called the *resampling* process.

After the surface mesh is created, it may contain topological and/or geometrical errors. These errors include isolated vertices (vertices not contained in a surface element), duplicated elements (surface elements that share the same vertices), or non-manifold vertices (a non-manifold vertex is one that is shared by two faces that do not share a common edge) (Figure 2.8) [65,66]. The re-meshed surface is repaired using embedded algorithms, which locate the

mentioned errors and delete them. This repair process is performed prior to and after the resampling process.

To produce a volume mesh in the third, and final, step, a point, P, is located inside the surface mesh. A 'distance-field' method is applied to ensure that this point lies inside the surface and not too close to the surface edge. In this method, the edge voxels, which define the surface, are assigned a value of '1' while assigning all other voxels a value of '0'. An iterative Gaussian smoothing kernel is then applied to create a 3D field in which the voxels are assigned numerical values that are inversely proportional to their corresponding distance from the surface edge. The coordinates of the voxel with the smallest field value are assigned to point P. Finally, tetrahedral elements are generated from point P towards the surface mesh to create a volume mesh of the 3D image that are created according to the *isovalues* and *opt* input parameters. Figure 2.9 shows a surface mesh converted into a volume mesh using the iso2mesh algorithm. The functions of the iso2mesh algorithm are shown in Appendix G.

Figure 2.8: Topological errors that may arise during the meshing process include (a) isolated, and (b) non-manifold vertices

Figure 2.9: Illustration of (a) surface mesh and (b) corresponding volume mesh as output by the iso2msh algorithm. The hollow surface mesh is filled with elements (highlighted in orange)

2.3.3.4 Meshing Using the iso2msh Algorithm

The final output of the iso2msh algorithm included two arrays for each model; a “nodes” array, which consisted of a list of all the nodes in the model along with their corresponding coordinates, and an “elements” array,

which listed the 4-node tetrahedral elements in the model along with the nodes associated with them. The resulting node and element definitions were then written into an Abaqus input file (.inp) using user-defined Matlab code, shown in Appendix G. This step was necessary in order to import the mesh into Abaqus CAE.

The element size and number of elements were controlled by changing the two parameters, *isovalues* and *opt*, in the iso2mesh algorithm. Changing these parameters allowed for the generation of coarse and fine meshes when optimizing the mesh for the finite element analysis, a processes discussed in Section 2.6.

2.4 Experimental testing

For the experimental analysis, a displacement controlled three-point bending test was performed on each of the seven bone specimens. The reaction force on each specimen was measured as displacement was applied. The force and displacement data was later used for the finite element analysis, a process described in more detail in Section 2.5. Details of the experimental setup are described below.

2.4.1 Setup and Calibration

The apparatus used in the test included a three-point bending frame, a load frame, a 5N load cell, a cantilever displacement measuring device (CDMD), and a P3 digital strain gauge indicator (see Appendix E). The deflection of the loading arm measured using the MTS Flextest built-in linear variable differential transformer (LVDT). The readings from the LVDT, however, were confounded by the presence of a localized deformation at the point of load application and would not correspond to the deflection of the specimens. Hence, deflection was measured using a cantilever displacement measuring device (CDMD) which provided a more accurate reading of the deflection due to bending only.

The CDMD was fabricated out of a spring steel beam and a strain gauge (Figure 2.10). The strain gauge was fixed to the end of the beam and measured strain at the tensile surface the beam. The deflection of the beam could be determined as it is proportional to the surface strain measured by the strain gauge [67]. Calibration of the CDMD as well as the load cell was required to ensure accurate measurements by the components were achieved.

Before testing the bone specimens, the load cell and strain gauge indicator were calibrated. The calibration process for both the load cell and CDMD are outlined in the section below.

Figure 2.10: Image of CDMD showing location of strain gauge

2.4.1.1 Calibration of the CDMD

The calibration process used to calibrate the CDMD involved applying a known deflection to the end of the CDMD beam while recording the voltage change of the strain gauge on the beam surface. The deflection applied was measured using a micrometer and the voltage change was measured using a P3 strain gauge indicator [Vishay Micro-Measurements, Raleigh, NC, USA]. This procedure was performed for a deflection range of 0mm - 0.6mm at 0.05mm intervals and a calibration graph for the “applied deflection” vs “strain gauge voltage reading” was created. The calibration process was performed twice, both before starting and after the completion of the three point bending test. A regression line was created from the calibration data of the CDMD from both calibrations. The calibration equations of the lines before and after testing along with the corresponding r^2 and squared error values are shown in Table 2.3. The

calibration data and resulting graph and regression line from the CDMD calibration are shown in Appendix C.

Table 2.3: Calibration equations along with r^2 and squared error values for calibration lines. In the equations, X represents the displacement measurement of the CDMD in mm, and ϵ represents the voltage measurement of the CDMD's strain gauge in volts.

	$X = 12.16\epsilon - 0.0039$	0.9995	0.008 mm
	$X = 12.10\epsilon - 0.0004$	0.9996	0.008 mm

2.4.1.2 Calibration of the load cell

A MTS [MTS Systems Corporation, Eden Prairie, MN] load cell with a capacity of 5N was used to measure the reaction force during the analysis. The load cell used was designed with built in shunt resistors used to automatically calibrate the device using the Flextest software. According to the specification sheet provided by MTS, the error associated with the load cell is 0.05%. Prior to testing the load cell was calibrated using this automatic calibration process.

2.4.2 Experimental test

To experimentally determine the structural modulus of elasticity of the bone specimens, a miniature three-point bending frame was used (see Figure 2.11). The bone specimen being tested rested on cylinders A and B of the frame,

both having a diameter of 1.5 mm, with a span of 7.8 mm kept constant for all the specimens (aspect ratios of specimens in Table 2.2).

A loading arm with a third identical cylinder, cylinder C, was then driven down by a Sintech 1/G servo electric MTS load frame [MTS Systems Corporation, Eden Prairie, MN] at a speed of 0.25 mm/s to bend the bone specimen to fracture. The loading procedure used to control the test was written in MTS Flextest, the controller software for the MTS load frame. The step definition of the test is shown in Appendix B.

To measure the reaction force on the top cylinder, an MTS 5N load cell was used (see Appendix E). The load cell was attached to the load frame between cylinder C and the driving shaft. Throughout the test, values for displacement and corresponding reaction forces were recorded by the MTS Flextest software at a rate of 20 Hz. During the test, the deflection of the specimen was measured using the CDMD. A measured reaction force of 0.01 N was applied by the CDMD. This force was corrected for after the test.

(a)

(b)

(c)

Figure 2.11: Three point bending test setup (a) showing dimensions. During the test, (b) cylinder C was driven down to bend the bone specimen. The CDMD (c) measures the deflection of the bone specimen as it is being bent

2.5 Finite Element Analysis

A finite element analysis was performed on each specimen model using Abaqus CAE v6.10-2 [Simulia, Providence, RI]. The analysis simulated the experimental three-point bending test discussed in Section 2.4, and was performed to determine an effective isotropic tissue modulus (E_{eff}) for each specimen. The models developed, using the procedure outlined in Section 2.3, were verified then imported into Abaqus CAE and used for the analysis. Three rigid cylinders were created in the assembly to simulate the three supports of the three point bending test. Simulating the supports using rigid cylinders and contact elements allowed for a more realistic load distribution on the specimens.

2.5.1 Input File

Abaqus input files ('.inp'), containing only geometry information, were written for each model using data obtained from the iso2mesh algorithm. These '.inp' files each contained node and element information for the corresponding model. Each node was defined by a node number, and three x, y, and z coordinates. Each element was defined by an element number and a list of all nodes associated with that element. The input file was then imported into Abaqus CAE, which was used to perform the finite element analysis.

2.5.2 FEA preprocessing

To perform a finite element analysis, an assembly was created to simulate the three-point bending test. The assembly consisted of the meshed bone model and three cylindrical rigid bodies simulating the contact points in the three-point bending test. The meshed model was imported into Abaqus CAE using the input file described above. The elements used in the meshed model were 4 node tetrahedral elements (C3D4).

2.5.2.1 Parts and Assembly

The rigid cylinders were defined using the Abaqus CAE preprocessor as analytical rigid bodies. By approximating the cylinders as totally rigid, a lower simulation time (approximately half) was achieved. This approximation is justified by the large difference in stiffness between the steel cylinders and bone, and was further verified as shown in validation Section 2.6.5. The diameter of the cylinder was defined as 1.5 mm (identical to the cylinders in the experimental analysis). The length of the cylinder was defined as 3mm, just wider than the bone specimens. The parts were positioned to simulate the experimental test, and contact surfaces between bone and cylinders were defined. The span between the two bottom cylinders matched that in the three point bending test at 7.8 mm. The final assembly is shown in Figure 2.12.

Figure 2.12: Assembly used in finite element analysis

2.5.2.2 Material Properties

The next step consisted of defining the material assigned to the specimen model. A linear isotropic model, defined by modulus of elasticity and Poisson's ratio was used. Poisson's ratio of bone was assumed to be 0.3 as verified by Ito et al. [39] and the modulus of elasticity for each specimen was initially defined as 1GPa. The rigid cylinders required no material definition as they would not deform during the simulation.

2.5.2.3 Boundary Conditions, Interactions and Applied Deflections

The constraints of the simulation included boundary conditions, interactions (contact definition), and applied deflections. The initial boundary conditions were defined for the specimen model and each of the three rigid cylinders. The specimen model was restricted at two points on the line of contact of the top cylinder in both the x and z direction as shown in Figure 2.13. This

constraint restricted the specimen from slipping or rotating and ensured bending in the y direction during the simulation. The two bottom cylinders were fully constrained in all directions. The top cylinder was constrained to prevent translation in the x and z directions and rotation about any axis.

Figure 2.13: Boundary conditions applied to finite element model shown (a) in the y-z plane and (b) in the x-z plane

The *general contact* algorithm in Abaqus Standard algorithm was used to define contact between the cylinders and the bone model, which introduced nonlinearity to the analysis. This contact algorithm used the *finite sliding* formulation, where arbitrary relative motion of the contact surfaces is permitted,

and 'surface to surface' formulation, where contact conditions are enforced over the entire contact region, during the analysis [68]. Each contact interaction occurred between two surfaces, a *master surface* and a *slave surface*, which resulted in a total of six contact surfaces (Figure 2.14). For general contact, any rigid surface associated with contact should be defined as the master surface and the deformable surface defined as the slave surface, hence the cylinder surfaces were defined as the master surfaces and the corresponding surfaces on the mesh as the slave surfaces. Contact constraints were enforced using the *penalty method*, in which slight penetration of the master surface into the slave surface, linearly proportional to the contact force, was permitted between the contacting surfaces based on a recommended stiffness factor of 10 times the underlying element stiffness [68]. Enforcing contact constraints using the penalty method ensured no localized high stresses formed between the contact surfaces.

Figure 2.14: Contact surfaces in the finite element model indicated in red.

2.5.3 FEA analysis (post-processing)

Figure 2.15a shows the final assembly with one of the specimens surrounded by three rigid cylinders prior to running the simulation. A displacement was applied to the top cylinder and was determined from the linear region of the resulting force displacement graphs of the three-point bending tests. The top cylinder would, hence, interact with the specimen model to deflect it. The simulated reaction force resulting between the top cylinder and bone was then compared to that measured during the experimental analysis, and the simulation was rerun iteratively with a different elastic modulus until the simulated reaction force matched experimental results with an elastic modulus value within 100 MPa. This elastic modulus corresponded to the effective isotropic tissue modulus (E_{eff}) of the bone. To avoid distorting elements during this forced contact procedure, the deflection was applied gradually with time (Appendix F). Figures 2.15a and 2.15b show the bone specimen before and after the simulation. The nonlinear analysis, due to nonlinear contact in the model, was solved using the Newton-Raphson method in Abaqus Standard. The average running time for each simulation was 300 minutes.

Figure 2.15: Showing Specimen 1 (a) before and (b) after simulating three point bending

2.6 FEM VALIDATION

To verify the results from the FEA simulations, a sensitivity analysis was performed for a number of parameters; namely, a mesh validation, the thresholding level applied to the μ CT scans, the type and number of the mesh elements used to create the models, and the use of rigid cylinders in the FEA model. The results of this analysis are shown below.

2.6.1 Volume Mesh Validation

Before proceeding with the analysis, the meshed models were verified by directly aligning the μ CT scans and volume mesh of each specimen in Amira 5.3.3 [Amira Visage Inc, Berlin, Germany], (Figure 2.16). The alignment was performed by importing the scans and volume of each model into the software and applying the appropriate translation and rotation transformations. The aligned model data for the specimens are shown in Appendix A.

Figure 2.16: Micro-CT scans and meshed volume of Specimen 1 aligned in Amira 5.3.3.

2.6.2 Threshold level validation

The threshold level used for the segmentation of the μ CT scanned images prior to reconstructing the model has a visual effect on the shape of the trabecular structure of the model. To ensure that the results of the simulation were not significantly affected by the threshold level, a sensitivity analysis was performed using models created with different threshold levels. Threshold levels with 8-bit grayscale intensity ranges of 50 - 70, 60 - 80, and 70 - 90, iteratively chosen based on visual confirmation, were used in creating three different models, (Figure 2.17). The results from running these simulations are shown in Table 2.4. Based on these results it was determined that the thresholding level had negligible effect on the final results, hence the middle level of thresholding, with a grayscale intensity value of 60-80, was used.

(a)

(b)

(c)

Figure 2.17: Bone Specimen meshed using grayscale threshold ranges of
(a) 50-70 (b) 60-80 and (c) 70-90

Table 2.4: Simulation results for specimen 1 using different threshold ranges

Simulation results for specimen 1 using different threshold ranges			
300000	E = 2.0 GPa	E = 2.0 GPa	E = 2.0 GPa
650000	E = 2.3 GPa	E = 2.3 GPa	E = 2.3 GPa

2.6.3 Element type Validation

The element type used in the analysis was determined by running the simulation using first order (4-node) tetrahedral, and second order (10-node) tetrahedral elements and comparing the results. Typically, second order tetrahedra are preferred over first order tetrahedra in FE meshes due to their capability to simulate areas of high stresses. According to Cifuentes and Kalbag, however, using a large number of 4-node tetrahedral elements will eventually result in similar simulation results as when fewer 10-node tetrahedral elements are used [69]. In modeling cancellous bone, it was important to accurately model the complex geometry of the trabecular tissue in order to obtain accurate results. To properly define this geometry, a large number of elements was required. With this high number of elements there was no difference in E_{eff} when 4-node or 10-node elements were used (Table 2.5). Therefore first order tetrahedra were used for the analysis. The results from the sensitivity analyses are outlined in Table 2.6

Table 2.5: Simulation results for specimen 1 using both 4-node and 10-node tetrahedral elements

Simulation Results for Specimen 1				
	E (GPa)	Run Time (min)	E (GPa)	Run Time (min)
300000	E = 2.0 GPa	50	E = 2.3 GPa	2100
650000	E = 2.3 GPa	150	E = 2.3 GPa	6900

2.6.4 Element Size Validation

When creating each mesh using the iso2mesh algorithm, the element size used was determined based on the algorithm's input parameters, *isovalues* and *opt* (Section 2.3.3). To verify the element size used in the meshes, a mesh convergence test was performed by initially creating a mesh discretization and recording the E_{eff} value obtained using that mesh. This procedure was then repeated for a finer mesh (approximately twice the number of elements) and the E_{eff} from both meshes were compared. Figure 2.18 shows the sensitivity of the element size on the mesh quality of the models. Table 2.5 shows the results of this process for specimen 1. The percentage error value was calculated using the following equation,

$$\%error = \frac{|E_{eff} \text{ previous mesh} - E_{eff} \text{ current mesh}|}{E_{eff} \text{ previous mesh}} \times 100\% \quad Eq. 2.1$$

A solution was obtained once an increase in the number of elements resulted in an error of less than 10% of the previous E_{eff} value. The iso2mesh input variables used to create element size of the solution were used in meshing the remaining models. Figure 2.19 shows the resulting convergence graph. The x-axis indicate the number of elements used in the mesh and the y-axis is the percent error calculated using Equation 2.1. The convergence criteria for determining the mesh size of the models resulted in an uncertainty of about ± 200 MPa in the final E_{eff} value.

Figure 2.18: Varying degree of detail as a result of the number of elements in the mesh.
Figure shows specimen 1 with (a) 300,000 (b) 650,000 and (c) 1,250,000 elements

Table 2.6: Results from convergence test including % error value calculated according to Equation 2.1

75,000	1.1	-
150,000	1.6	45.5%
300,000	2.0	25.0%
650,000	2.3	15.0%
1,250,000	2.5	8.7%

Figure 2.19: Convergence test performed for the finite element analysis of specimen 1. The curve shown represents the percent error associated with the number of elements used in the finite element model. The results indicate a difference of 8.7% in E_{eff} between using 650,000 elements and 1,250,000 elements

2.6.5 Rigid Cylinder Validation

In the analysis, analytical rigid cylinders were created to simulate the contact points of the three-point bend test. A verification analysis was performed using a beam specimen, assigned an elastic modulus of 2 GPa, and deformable steel cylinders, assigned an elastic modulus of 200 GPa, to validate choosing rigid cylinders, as opposed to deformable steel cylinders, in the simulation. The results of this analysis confirmed that using rigid cylinders in the final analysis would not affect the results.

2.7 Specimen Models

Obtaining the models for the computational analysis involved creating volume models from the μ CT scans, then meshing these models for the finite element analysis. The μ CT scans of each specimen were segmented and the reconstructed models were obtained using Matlab. The reconstructed models were meshed, using tetrahedral elements, to be used in a finite element analysis. The size of the elements, hence number of elements, used for each model was determined based on the convergence procedure outlined in Section 2.6. Table 2.7 shows the number of elements used to mesh each specimen, and Figures 2.20a to 2.20g show the individual meshed models.

Table 2.7: Number of elements used to mesh each specimen volume model

Specimen	Number of elements
1	731902
2	865791
3	592896
4	765973
5	795633
6	763280
7 (Control)	671810

(a)

(b)

(c)

(d)

(e)

(f)

(g)

Figure 2.20: Meshed models of specimens (a) 1, (b) 2, (c) 3, (d) 4, (e) 5, (f) 6, and (g)

Chapter 3

Results

The results (namely the elastic moduli of the specimens) obtained from the analysis allowed for a primary comparison between FAI bone and healthy bone. The goal was to gain a better understanding of the relationship between subchondral bone properties in FAI in the region of the bump. The mechanical properties of six FAI specimens and a healthy (control) specimen were obtained through a combined (experimental and finite element) analysis. The results from both parts of the analysis for all seven specimens are outlined in this chapter.

3.1 Porosity of the Specimens

The porosity values for each of the specimens are outlined in Table 3.1. As shown, these values range between 35% and 78% which is typical for trabecular bone in human femurs [25].

Table 3.1: Porosity values of the bone specimens

1	35%
2	49%
3	40%
4	78%
5	59%
6	61%
7 (Control)	64%

3.2 Experimental Results

The experimental analysis involved each of the bone specimens undergoing a three-point bending test until fracture (except for specimens 1 and 6 which did not fracture). The data collected during the tests included force data from the load cell and displacement data from the CDMD. For each specimen, a force vs displacement graph was created using the data obtained during the test, Figure 3.1. The initial linear portion of each graph indicating elastic deformation of the specimen during the test and was the region of interest in this analysis. Beyond that region, nonlinear behavior, due to collapsing within the trabecular structure and plastic deformation, occurred followed by failure. The slope of the linear portion of the graphs were calculated as the stiffness values of the corresponding specimens. A point, P, at 80% of the displacement prior to

yielding, was chosen within the linear elastic region of each graph. The stiffness value of each specimen was calculated as the slope of the linear region. An estimated structural modulus $E_{\text{structural}}$ of elasticity was calculated using simple beam theory, where the specimens were assumed to be perfect beams and shear effects were not accounted for (Equations shown in Appendix D). Calculating $E_{\text{structural}}$ was done to verify its correlation with porosity. A porosity vs $E_{\text{structural}}$ is shown in Figure 3.2. The displacement and corresponding reaction force values from point P, along with the stiffness and $E_{\text{structural}}$ values are shown in Table 3.2. The data from each point was then analyzed in the finite element analysis to determine the effective tissue modulus. The results from the finite element analysis are shown in the next section.

Table 3.2: Total elastic displacement and corresponding reaction force of each specimen during three point bending

1	0.05	4.3	89.6	2.5
2	0.08	1	12.5	0.3
3	0.04	1.1	27.5	1.0
4	0.08	0.7	8.8	0.2
5	0.06	1	16.7	0.2
6	0.06	1	16.7	0.2
7 (Control)	0.04	1.1	27.5	0.4

Figure 3.1: Graphs for reaction force vs. displacement from three-point bending of the specimens

3.3 Finite Element Analysis Results

After completing the finite element analysis E_{eff} values for the specimens were obtained. The results from the FEA are shown in Table 3.3. A porosity vs E_{eff} graph is shown in Figure 3.3. The E_{eff} obtained for the six FAI affected specimens varied between 1.2 GPa and 2.5 GPa. The E_{eff} of the control specimen was 1.0 GPa.

Table 3.3: The effective isotropic tissue modulus of each of the seven specimens as obtained from the finite element analysis

1	731902	150	2.3
2	865791	400	1.2
3	592896	300	2.5
4	765973	350	2.2
5	795633	200	1.5
6	763280	400	2.1
7 (Control)	671810	250	1.0

Figure 3.2: Graph of porosity vs $E_{\text{structural}}$

Figure 3.3: Graph of porosity vs E_{eff}

Chapter 4

Discussion

The importance of the results obtained from this analysis along with their potential applications are discussed in the following section. The limitations of the analysis along with recommendations for future work related to this study are also included.

4.1 The Relationship Between Osteoarthritis and FAI

This study determined effective isotropic tissue modulus of subchondral bone in 6 symptomatic cam type FAI patients. Although no general conclusion can be drawn due to the insufficient number of control specimens available, the subchondral bone E_{eff} values in the FAI patients were almost twice that of the healthy control specimen. Since the mineral content of bone is the determining factor for elastic modulus, an increase in E_{eff} value suggests that there is a substantial amount of increased mineralization present in the 6 FAI bone specimens. This increased tissue level modulus with increased density and mineralization may contribute to the degeneration cascade of the the articular cartilage.

The reason for increased mineralization of the bone specimens is not known, but it has been postulated that mineralization can be a result of contact induced microfractures of the subchondral layer. According to Simon et al., microfractures in the trabecular structure of subchondral bone can occur as a result of excessive loading expected during FAI. During the healing process, a higher mineralization was observed in the trabecular tissue [6].

4.2 Experimental and FEA Results

The E_{eff} of specimens 2 and 5 were shown to be significantly lower than the other 4 FAI specimens. A reason for the lower E_{eff} might be due to active remodeling, an ongoing process where bone cells are removed (bone resorption) and replaced by newly formed bone in response to local stresses and/or other environmental factors [25, 70]. Active bone remodeling has been observed at the bump of FAI patients and is known to decrease the tissue level modulus of bone by lowering its calcified mineral content [71,72].

The porosity vs $E_{\text{structural}}$ graph indicates that a correlation may exist between these two properties (Figure 3.2). This possible correlation may be due to the fact that the $E_{\text{structural}}$ of bone relies on its trabecular structure as well as its tissue properties. Looking at the porosity vs E_{eff} graph, however, no clear correlation is observed since E_{eff} quantifies the elastic properties of the bone tissue and is unaffected by the structure, or porosity, of the bone (Figure 3.3).

The $E_{\text{structural}}$ values obtained were significantly lower than E_{eff} (between 0.5 and 0.1 times that of E_{eff} depending on the specimen) due to the porosity of the structure.

4.3 Error Analysis

An error analysis was performed to quantify the effect of any sources of error throughout the analysis. The sources of error included those due to measuring the specimen displacement in the experimental analysis, measuring the dimensions of each specimen, and neglecting shear in the $E_{\text{structural}}$ calculations.

The displacement measurements obtained from the CDMD did not take into account the vertical parallel shift of the specimen due to local deformations to the specimen from the two support cylinders. To quantify the error associated with the CDMD measurements, the displacement of the top cylinder (measured by the LVDT), which included the displacement due to bending of the specimen and the local deformations due to the support cylinders, was compared with the measurements by the CDMD, which only included the displacement due to bending of the specimen and the local deformation of the two button support cylinders. An averaged difference of 25% at the point used in the finite element analysis was found between the displacement values, which

corresponded to localized deformation of the bone specimen at the top cylinder. This value was calculated as the ratio of the difference between the two displacement measurements to the total deflection measured by the CDMD. Assuming the loads are symmetrically distributed half of this error existed at each of the bottom cylinders, the error associated with the CDMD was determined to be 13%. Figure 4.1 shows the load displacement graphs of both displacement measuring devices from one of the tests. This error in displacement measurement corresponded to an overestimation of up to 0.01 mm, which resulted in an error of ± 100 MPa in the E_{eff} value.

Other sources of error were shown to exist in the $E_{\text{structural}}$ calculations. As previously mentioned, shear effects were neglected when calculating $E_{\text{structural}}$. Due to the relatively small aspect ratio of the three point bending test, shear effects were not negligible and according to Schriefer et al., a 35 % error was found to exist when bending bone at an aspect ratio of 8 [44]. As well, due to the varying cross sectional area and porosity of the specimens, the second moment of area could not be accurately calculated. The $E_{\text{structural}}$, however, is an approximated value calculated to verify a relationship between the porosity of the specimens and $E_{\text{structural}}$.

Figure 4.1: Graph of force vs displacement for specimen 4 using both the LVDT and CD measure displacement

4.4 Limitations

The results obtained were based on six FAI specimens and one control. This sample size was limited by the number of accessible patients. A larger sample size, particularly for control specimens, is necessary to draw any conclusive data from this study.

The density of the bone tissue was assumed to be constant and an effective modulus of elasticity of the tissue was obtained. The variability of the

density in the tissue was, hence, not accounted for and might have limited the certainty of the results.

4.5 Future Work

More samples are required to confirm the findings of this investigation. Recently, 5 additional FAI specimens along with an additional control specimen have been obtained for this study. These specimens will undergo the procedure outlined in this thesis to obtain their corresponding E_{eff} . Increasing the sample size (n) will further validate the increase in E_{eff} due to FAI.

A quantitative micro-CT scan can also be performed to obtain the density distribution of the specimen models. Assigning density values to the individual elements of the model would be possible and allow for the E_{eff} distribution within the specimen to be determined.

One speculation made based on the results of this analysis was the increased mineralization of the FAI specimens. This speculation can be further analysed by determining the mineral content of the FAI specimens using an ash density [73] or spectroscopic [74] analysis.

Chapter 5

Conclusion

5.1 Conclusions

The resulting E_{eff} values varied between 1.2 GPa and 2.3 GPa for the FAI specimens. The E_{eff} of the healthy bone specimen had a value of 1.0 GPa. Based on the limited number of specimens in this analysis, the results suggested that subchondral bone from cam-type FAI patients possess an increased stiffness due to FAI. An increased stiffness of the subchondral bone in the femoral head has been speculated to promote the notion of OA [6,15,75], which is an indication that FAI may be considered an early symptom of OA.

The porosity of the specimens and the E_{eff} values obtained suggest that the trabecular structure of bone does not affect its tissue properties. Hence,, no correlation was observed between the two variables. Relating the specimen porosity to the corresponding $E_{\text{structural}}$ values a negative correlation was found.

The results obtained in this thesis are primary and may not be considered representative due to the lack of control specimens analyzed. These results, however, can be used to guide future analyses and with additional control specimens, general, more representative results may be obtained.

5.2 Contributions

As part of the experimental analysis, a three point bending test was performed on human biopsies obtained from the femoral hips of FAI patients. The development, manufacturing, and calibration of a displacement measuring device was achieved and used in measuring the deflection of the specimens in the experimental analysis.

Micro finite element models of the specimens were developed in the analysis to simulate the trabecular structure of the specimens. These models were used in a finite element analysis to determine the tissue level properties of 6 subchondral bone specimens from FAI patients and 1 control specimen from a hip showing no signs of FAI.

The tissue level properties of subchondral bone in FAI patients were quantified for the first time in this analysis. These results provided material property data for patient specific finite element models which will determine the acetabular subchondral bone and cartilage stresses. This will help understand whether FAI related OA is due to biomechanical effects.

References

- [1] Osteoporosis Canada "Osteoporosis and Osteoarthritis" [WebPage].
Available: [http:// www.osteoporosis.ca/index.php/ci_id/5507/la_id/1.htm](http://www.osteoporosis.ca/index.php/ci_id/5507/la_id/1.htm)
- [2] Statistics Canada, Canadian Community Health Survey, 2008
- [3] Lawrence, R. C., Felson, D. T., Helmick, C. G., Arnold, L. M., Choi, H., Deyo, R. a, Gabriel, S., et al. (2008). Estimates of the prevalence of arthritis and other rheumatic conditions in the United States. Part II. *Arthritis and rheumatism*, 58(1), 26-35
- [4] Matsui, H., Shimizu, M., & Tsuji, H. (1997). Cartilage and subchondral bone interaction in osteoarthrosis of human knee joint: a histological and histomorphometric study. *Microscopy research and technique*, 37(4), 333-42.
- [5] Radin, E. L., Paul, I. L., & Rose, R. M. (1972). Role of mechanical factors in pathogenesis of primary osteoarthritis. *Lancet*, 1(7749), 519-22.
- [6] Simon, S. R., Radin, E. L., Paul, I. L., & Rose, R. M. (1972). The response of joints to impact loading - II In vivo behavior of subchondral bone, *J. biomechanics*, 5(3), 267-72.
- [7] Tanzer, M., & Noiseux, N. (2004). Osseous Abnormalities and Early Osteoarthritis. *Clinical Orthopaedics and Related Research*, 429(429), 170-177.
- [8] Ganz, R., Parvizi, J., Beck, M., Leunig, M., Nötzli, H., & Siebenrock, K. a. (2003). Femoracetabular impingement: a cause for osteoarthritis of the hip. *Clinical orthopaedics and related research*, (417), 112-20.

- [9] Beck, M., Kalhor, M., Leunig, M., & Ganz, R. (2005). Hip morphology influences the pattern of damage to the acetabular cartilage: femoroacetabular impingement as a cause of early osteoarthritis of the hip. *The Journal of bone and joint surgery. British volume*, 87(7), 1012-8.
- [10] Leunig, M., Beaulé, P. E., & Ganz, R. (2009). The concept of femoroacetabular impingement: current status and future perspectives. *Clinical orthopaedics and related research*, 467(3), 616-22.
- [11] Adolphe Braun, (2010). "Imp Braun Genève" [WebPage]. Available: <http://www.4shared.com/photo/vwGEUnvm/skeleton-9340.html?cau2=403tNull>
- [12] S. Standring, *Gray's Anatomy The Anatomical Basis of Clinical Practice* 39th ed., Elsevier, Churchill Livingstone, © 2005
- [13] Image generated using 3D4Medicals Skeleton System Pro III App
- [14] M. Nordin and V. H. Frankel, *Basic Biomechanics of the Musculoskeletal System* 3rd ed., Lippincot Williams & Wilkins, © 2001
- [15] Westacott, C. (2002). Interactions between subchondral bone and cartilage in OA. Cells from osteoarthritic bone can alter cartilage metabolism. *Journal of musculoskeletal & neuronal interactions*, 2(6), 507-9.
- [16] Columbia University Medical Center "Anatomy of the Joint" [WebPage]. Available: http://www.cumc.columbia.edu/dept/rehabmusculoskeletal_health/anatomy.html.gif [2012]
- [17] Costa, M. L., Achten, J., Parsons, N. R., Edlin, R. P., Foguet, P., Prakash, U., & Griffin, D. R. (2012). Total hip arthroplasty versus resurfacing

- arthroplasty in the treatment of patients with arthritis of the hip joint: single centre, parallel group, assessor blinded, randomised controlled trial. *BMJ* (Clinical research ed.), 344
- [18] Kavanagh, B. F., Ilstrup, D. M., & Fitzgerald, R. H. (1985). Revision total hip arthroplasty. *The Journal of bone and joint surgery. American volume*, 67(4), 517-26.
  - [19] Siebenrock, K. A., Schoeniger, R., & Ganz, R. (2003). Anterior femoro-acetabular impingement due to acetabular retroversion. Treatment with periacetabular osteotomy. *The Journal of bone and joint surgery. American volume*, 85-A(2), 278-86.
  - [20] Siebenrock, K. A., Leunig, M., & Ganz, R. (2001). Periacetabular osteotomy: the Bernese experience. *The Journal of Bone and Joint Surgery*, 2001; 83:449-449 50(9)
  - [21] Reynolds, D., Lucas, J., & Klaue, K. (1999). Retroversion of the acetabulum. A cause of hip pain. *The Journal of bone and joint surgery. British volume*, 81(2), 281-8
  - [22] Kevin Neeld Athletic Development, LLC © 2007 - 2012 "Types of FAI" [Webpage] Available: <http://www.kevinneeld.com/2011/training-around-femoroacetabular-impingement> [2012]
  - [23] [2012] Marcus J K Bankes © 2011 [Webpage] Available: http://www.bankes.com/hipsurgery_2011/hip-arthroscopy-sports-hip-injuries-and-femoroacetabular-impingement/femoroacetabular-impingement.html [2012]

- [24] C. Ross Ethier and Craig A. Simmons, *Introductory Biomechanics From Cells to Organisms*, Cambridge University Press, © 2007
- [25] Van C. Mow and Rik Huiskes, *Basic Orthopaedic Biomechanics and Mechano- Biology* 3rd ed., Lippincot Williams & Wilkins, © 2005
- [26] SEER Training Modules, Cancer Registration & Surveillance Module. U. S. National Institutes of Health, National Cancer Institute. [Webpage]
Available: <http://training.seer.cancer.gov/anatomy/skeletal/tissue.html> [15 April 2012]
- [27] Henry Gray, *Anatomy of the Human Body*, 20th ed. Philadelphia: Lea & Febiger, 1918; Bartleby.com, 2000. www.bartleby.com/107/. [2012]
- [28] Bayraktar, H. H., Morgan, E. F., Niebur, G. L., Morris, G. E., Wong, E. K., & Keaveny, T. M. (2004). Comparison of the elastic and yield properties of human femoral trabecular and cortical bone tissue. *Journal of Biomechanics*, 37(1), 27-35.
- [29] L. Gibson and M. Ashby, *Cellular Solids Structure & Properties*, Pergamon Press, © 1988
- [30] Whitehouse, W. J., & Dyson, E. D. (1974). Scanning electron microscope studies of trabecular bone in the proximal end of the human femur. *Journal of anatomy*, 118(Pt 3), 417-44.
- [31] Thompson, M. S., Flivik, G., Juliusson, R., Odgaard, A., & Ryd, L. (2004). A comparison of structural and mechanical properties in cancellous bone from the femoral head and acetabulum. *Proceedings of the Institution of*

Mechanical Engineers. Part H, Journal of engineering in medicine, 218(6), 425-9

- [32] Gibson, L. J. (1985). The mechanical behaviour of cancellous bone. Journal of Biomechanics, 18(5), 317-328.
- [33] Hamed, E., Jasiuk, I., Yoo, A., Lee, Y., & Liszka, T. (2012). Multi-scale modelling of elastic moduli of trabecular bone. Journal of the Royal Society, Interface / the Royal Society, (January).
- [34] Burstein, A. H., Zika, J. M., Heiple, K. G., & Klein, L. (1975). Contribution of collagen and mineral to the elastic-plastic properties of bone. The Journal of bone and joint surgery. American volume, 57(7), 956-61.
- [35] Hsieh, C. L., Tuan, W. H., & Wu, T. T. (2004). Elastic behaviour of a model two- phase material. Journal of the European Ceramic Society, 24(15-16), 3789-3793.
- [36] Lenthe, G. H., Voide, R., Boyd, S. K., & Müller, R. (2008). Tissue modulus calculated from beam theory is biased by bone size and geometry: implications for the use of three-point bending tests to determine bone tissue modulus. Bone, 43(4), 717-23.
- [37] Ladd, a J., & Kinney, J. H. (1998). Numerical errors and uncertainties in finite- element modeling of trabecular bone. Journal of biomechanics, 31(10), 941-5.
- [38] Keaveny, T. M., Pinilla, T. P., Crawford, R. P., Kopperdahl, D. L., & Lou, a. (1997). Systematic and random errors in compression testing of trabecular

- bone. *Journal of orthopaedic research : official publication of the Orthopaedic Research Society*, 15(1), 101-10.
- [39] Ito, M., Nishida, a, Koga, a, Ikeda, S., Shiraishi, a, Uetani, M., Hayashi, K., et al. (2002). Contribution of trabecular and cortical components to the mechanical properties of bone and their regulating parameters. *Bone*, 31(3), 351-8.
  - [40] Turner, C. H., & Burr, D. B. (1993). Basic biomechanical measurements of bone: a tutorial. *Bone*, 14(4), 595-608.
  - [41] Brown, T. D., & Ferguson, a B. (1980). Mechanical property distributions in the cancellous bone of the human proximal femur. *Acta orthopaedica Scandinavica*, 51(3), 429-37.
  - [42] Zysset, P. K., Guo, X. E., Hoffler, C. E., Moore, K. E., & Goldstein, S. A. (1999). Elastic modulus and hardness of cortical and trabecular bone lamellae measured by nanoindentation in the human femur. *Journal of biomechanics*, 32(10), 1005-12.
  - [43] Leppänen, O., Sievänen, H., Jokihaara, J., Pajamäki, I., & Järvinen, T. L. N. (2006). Three-point bending of rat femur in the mediolateral direction: introduction and validation of a novel biomechanical testing protocol. *Journal of bone and mineral research : the official journal of the American Society for Bone and Mineral Research*, 21(8), 1231-7.
  - [44] Schriefer, J. L., Robling, A. G., Warden, S. J., Fournier, A. J., Mason, J. J., & Turner, C. H. (2005). A comparison of mechanical properties derived

- from multiple skeletal sites in mice. *Journal of biomechanics*, 38(3), 467-75.
- [45] Gustafson, M. B., Martin, R. B., Gibson, V., Storms, D. H., Stover, S. M., Gibeling, J., & Griffin, L. (1996). Calcium buffering is required to maintain bone stiffness in saline solution. *Journal of biomechanics*, 29(9), 1191-4.
  - [46] Choi, K., Kuhn, J. L., Ciarelli, M. J., & Goldstein, S. a. (1990). The elastic moduli of human subchondral, trabecular, and cortical bone tissue and the size-dependency of cortical bone modulus. *Journal of biomechanics*, 23(11), 1103-13
  - [47] Niebur, G. L., Feldstein, M. J., Yuen, J. C., Chen, T. J., & Keaveny, T. M. (2000). High-resolution finite element models with tissue strength asymmetry accurately predict failure of trabecular bone. *Journal of biomechanics*, 33(12), 1575-83.
  - [48] Boccaccio, A., Ballini, A., Pappalettere, C., Tullo, D., Cantore, S., & Desiate, A. (2011). Finite element method (FEM), mechanobiology and biomimetic scaffolds in bone tissue engineering. *International journal of biological sciences*, 7(1), 112-32
  - [49] Wolfram, U., Wilke, H.-J., & Zysset, P. K. (2010). Valid micro finite element models of vertebral trabecular bone can be obtained using tissue properties measured with nanoindentation under wet conditions. *Journal of biomechanics*, 43(9), 1731-7. Elsevier.
  - [50] Ladd, a J., Kinney, J. H., Haupt, D. L., & Goldstein, S. a. (1998). Finite element modeling of trabecular bone: comparison with mechanical testing

- and determination of tissue modulus. *Journal of orthopaedic research* : official publication of the Orthopaedic Research Society, 16(5), 622-8.
- [51] Bevill, G., & Keaveny, T. M. (2009). Trabecular bone strength predictions using finite element analysis of micro-scale images at limited spatial resolution. *Bone*, 44(4), 579-84. Elsevier Inc.
- [52] Bonse, U., Busch, F., Günnewig, O., Beckmann, F., Pahl, R., Delling, G., Hahn, M., et al. (1994). 3D computed X-ray tomography of human cancellous bone at 8 microns spatial and 10⁻⁴ energy resolution. *Bone and mineral*, 25(1), 25-38
- [53] Said, R., Chang, J., Young, P., Tabor, G., & Coward, S. (2008). Image-Based Meshing of Patient-Specific Data: Converting Medical Scans Into Highly Accurate Computational Models. 2008 2nd International Conference on Bioinformatics and Biomedical Engineering, 1672-1676.
- [54] Link, T. M., Majumdar, S., Lin, J. C., Newitt, D., Augat, P., Ouyang, X., Mathur, a, et al. (1998). A comparative study of trabecular bone properties in the spine and femur using high resolution MRI and CT. *Journal of bone and mineral research : the official journal of the American Society for Bone and Mineral Research*, 13(1), 122-32.
- [55] Ulrich, D., van Rietbergen, B., Weinans, H., & Regsegger, P. (1998). Finite element analysis of trabecular bone structure: a comparison of image-based meshing techniques. *Journal of biomechanics*, 31(12), 1187-92.

- [56] Cuadros-Vargas, a. J., Nonato, L. G., Minghim, R., & Etienne, T. (2005).
Imesh: An Image Based Quality Mesh Generation Technique. XVIII
Brazilian Symposium on Computer Graphics and Image Processing
(SIBGRAPI'05), 341-348.
- [57] Courchesne, O., Guibault, F., Dompierre, J., & Cheriet, F. (n.d.). Adaptive
Mesh Generation of MRI Images for 3D Reconstruction of Human Trunk,
1040-1051.
- [58] Fang, Q., & Boas, D. A. (2009). Tetrahedral mesh generation from
volumetric binary and grayscale images. 2009 IEEE International
Symposium on Biomedical Imaging: From Nano to Macro (pp. 1142-1145).
IEEE.
- [59] Spatz, H. C., O'Leary, E. J., & Vincent, J. F. (1996). Young's moduli and
shear moduli in cortical bone. Proceedings. Biological sciences / The
Royal Society, 263(1368), 287-94.
- [60] Pham, D. L., Xu, C., & Prince, J. L. (2000). Current methods in medical
image segmentation. Annual review of biomedical engineering, 2, 315-37.
- [61] Haralick, R. M., & Shapiro, L. G. (1985). Image segmentation techniques.
Computer Vision, Graphics, and Image Processing, 29(1), 100-132.
- [62] Glover, G. H. (1982). Compton scatter effects in CT reconstructions.
Medical physics, 9(6), 860-7.
- [63] Prince, Medical Imaging Signals and Systems Upper Saddle River, N.J.
Pearson Prentice Hall, c2006
- [64] McNitt-Gray, M. (2006). Tradeoffs in CT Image Quality and Dose.

- [65] John M. Lee, Introduction to Smooth Manifolds, Springer, 2003
- [66] Topology and Geometry of Manifolds, G. Matic, C. McCoy, Proceedings of Symposia in Pure Mathematics, Volume 71, 2001, Georgia International Topology Conference, University of Georgia
- [67] Richard G. Budynas, Advanced Strength and Applied Stress Analysis, McGraw - Hill, 1999
- [68] Simulia, Abaqus 6.10 Analysis User's Manual Volume 5: *Prescribed Conditions, Constraints & Interactions*, Dessault Systèmes, © 2010
- [69] A.O. Cifuentes, A. Kalbag, A performance study of tetrahedral and hexahedral elements in 3-D finite element structural analysis, Finite Elements in Analysis and Design, Volume 12, Issues 3–4, December 1992, Pages 313-318.
- [70] Hill, P. (1998). Bone remodelling. Journal of Orthodontics, 25(2), 101-107.
- [71] Gryn timer, M. D., Alpert, B., Katz, I., Lieberman, I., & Pritzker, K. P. (1991). Subchondral bone in osteoarthritis. Calcified tissue international, 49(1), 20-6.
- [72] Li, B., & Aspden, R. M. (1997). Mechanical and material properties of the subchondral bone plate from the femoral head of patients with osteoarthritis or osteoporosis. Annals of the rheumatic diseases, 56(4), 247-54.
- [73] Li. Mosekilde, Le. Mosekilde, C.C. Danielson, Biomechanical competence of vertebral trabecular bone in relation to ash density and age in normal individuals, J. Bone, Volume 8, Issue 2, 1987, Pages 79-85.

- [74] E. P. Paschalis, F. Betts, E. DiCarlo, FTIR Microspectroscopic Analysis of Normal Human Cortical and Trabecular Bone, *Calcified Tissue Int.*, Volume 61, Number 6, 1997, Pages 480-486
- [75] Dieppe, P., Cushnaghan, J., Young, P., & Kirwan, J. (1993). Prediction of the progression of joint space narrowing in osteoarthritis of the knee by bone scintigraphy. *Annals of the rheumatic diseases*, 52(8), 557-63.

Appendices

Appendix A: Mesh Validation: CT and Mesh Alignment

Appendix B: MTS Step Definitions

Appendix C: Calibration Data and Results

Appendix D: Estructural Calculations

Appendix E: Equipment and Apparatus

Appendix F: Time Step Increments for FEA

Appendix G: Matlab Algorithms

Appendix A: Mesh Validation: CT and Mesh Alignment

Figure A1: Specimen 1 Micro-CT scans and meshed volume aligned in Amira 5.3.3.

Figure A2: Specimen 2 Micro-CT scans and meshed volume aligned in Amira 5.3.3.

Figure A3: Specimen 3 Micro-CT scans and meshed volume aligned in Amira 5.3.3.

Figure A4: Specimen 4 Micro-CT scans and meshed volume aligned in Amira 5.3.3.

Figure A5: Specimen 5 Micro-CT scans and meshed volume aligned in Amira 5.3.3.

Figure A6: Specimen 7 Micro-CT scans and meshed volume aligned in Amira 5.3.3.

Figure A7: Specimen 7 (Control) Micro-CT scans and meshed volume aligned in Amira 5.3.3.

Appendix B: MTS Step Definitions

The image below is a screenshot from the MTS Flextest software showing the step definitions used in the experimental analysis

Figure B1: Step definition for experimental analysis as created in Flextest

Appendix C: Calibration Data and Results

The data and graphs from the CDMD calibration both before and after performing the experimental analysis are shown in Tables C1-C2 and Figures C1-C2.

	Strain Gauge Output (V)	Micrometer Reading (in)	Micrometer Reading (mm)	Standard error	Standard error squared
Before Test	0.000	0.000	0.000	4.25E-03	8.49E-03
	0.004	0.002	0.051		
	0.009	0.004	0.102		
	0.013	0.006	0.152		
	0.017	0.008	0.203		
	0.021	0.010	0.254		
	0.026	0.012	0.305		
	0.030	0.014	0.356		
	0.034	0.016	0.406		
	0.038	0.018	0.457		
	0.042	0.020	0.508		
	0.046	0.022	0.559		
	0.050	0.024	0.610		

Table C1: Voltage output from CDMD strain gauge as a result of displacement applied by micrometer (Before testing)

Figure C1: Pre-testing CDMD calibration graph showing CDMD strain gauge reading on the x-axis and its deflection as measured by the micrometer on the y-axis

	Strain Gauge Output (V)	Micrometer Reading (in)	Micrometer Reading (mm)	Standard error	Standard error squared
	0	0.000	0.000	3.97E-03	7.94E-03
	0.004	0.002	0.051		
	0.008	0.004	0.102		
	0.013	0.006	0.152		
	0.017	0.008	0.203		
After Test	0.021	0.010	0.254		
	0.025	0.012	0.305		
	0.030	0.014	0.356		
	0.034	0.016	0.406		
	0.038	0.018	0.457		
	0.042	0.020	0.508		
	0.046	0.022	0.559		
	0.050	0.024	0.610		

Table C2: Voltage output from CDMD strain gauge as a result of displacement applied by micrometer (After testing)

Figure C2: Post-testing CDMD calibration graph showing CDMD strain gauge reading on the x-axis and its deflection as measured by the micrometer on the y-axis

Appendix D: Equations used to calculate $E_{\text{structural}}$

$$M = \frac{l \cdot F}{4}$$

$$I_x = \frac{w \cdot t^3}{12}$$

$$\sigma = \frac{My}{I_x}$$

$$\varepsilon = \frac{y}{R}$$

$$E_{\text{Structural}} = \frac{\sigma}{\varepsilon}$$

Where,

w: Width of the specimen (mm)

t: Thickness of the specimen (mm)

A: Approximate cross sectional area of specimen (mm²)

M: Bending moment (Nmm)

l: Span of three point bending test (mm)

F: Reaction force on specimen (N)

I_x: Second moment of area (mm⁴)

y: Deflection of specimen (mm)

σ: Stress due to bending (MPa)

ε: Strain due to bending (μstrain)

R: Radius of curvature of the specimen (mm)

E_{structural}: Approximate structural modulus of elasticity (GPa)

Specimen	Thickness t (mm)	Width w (mm)	Approx. Porosity	Approx. Area (mm ²)	Displacement (mm)	Force (N)	Stiffness (N/mm)	Moment (Nmm)	Second Moment of Area (mm ⁴)	Stress (MPa)	Strain (μ strain)	Structural Young's Modulus (GPa)
1	1.4	2.4	0.35	3.36	0.048	4.3	89.6	8.4	0.55	0.74	0.3	2.5
2	1.6	1.7	0.49	2.72	0.08	1.0	12.5	2.0	0.58	0.27	0.8	0.3
3	1.3	2.3	0.4	2.99	0.04	1.1	27.5	2.2	0.42	0.20	0.2	1.0
4	1.6	1.9	0.78	3.04	0.08	0.7	8.8	1.4	0.65	0.17	0.8	0.2
5	1.9	2.3	0.59	4.37	0.06	1.0	16.7	2.0	1.31	0.09	0.5	0.2
6	1.7	2.5	0.61	4.25	0.06	1.0	16.7	2.0	1.02	0.11	0.5	0.2
Control	1.7	2.8	0.64	4.76	0.04	1.1	27.5	2.2	1.15	0.08	0.2	0.4

Table D1: Values obtained for Estructural calculations

Appendix E: Equipment and Apparatus

MTS Load Frame

MTS Sintech 1/G Mechanical Load Frame

PN: 200-1005

Vishay P3 Strain Gauge Indicator

Vishay Instruments Division

P-3500 Strain Indicator

MTS Load Transducer

MTS 5N Load Transducer

PN - 4501016/B

SN - 650840

Buehler Diamond tip Saw

Buehler Isomet Low Speed Saw

Cat # 11-1180-170

S/N 347-IS-8005

Appendix F: Abaqus Time Step Increments for Applying Displacement in FEA

Time In Step	Fraction of Applied Displacement
1	0.05
2	0.1
3	0.15
4	0.2
5	0.25
6	0.3
7	0.35
8	0.4
9	0.45
10	0.5
11	0.55
12	0.56
13	0.57
14	0.58
15	0.59
16	0.6
17	0.61
18	0.62
19	0.63
20	0.64
21	0.65
22	0.66
23	0.68
24	0.7
25	0.72
26	0.74
27	0.76
28	0.78
29	0.8
30	0.9
31	1

Table F1: Change in fraction of applied displacement with FEA step time

Appendix G: Matlab Algorithms

The main functions used from Fang and Boas's iso2mesh algorithm can be obtained from the iso2mesh webpage, <http://iso2mesh.sourceforge.net/cgi-bin/index.cgi>. The functions written for the analysis are outlined below.

Mesh

```
function
[I,II,III,node,elem,face]=Mesh(iso,opt,maxvol,plot)
%Function
%Mesh: used to convert a series of image slices into a
single meshed
%      volume

%Output Arguments
%node: array containing the x, y, and z coordinates of the
nodes
%elem: array containing the node numbers defining the
elements
%face: array containing node indices of the triangular
surfaces

%Input Arguments
%iso:          a list of the isovalues where the
levelset is defined
%opt:          size of Delaunay sphere (initial elemtn
size)
%maxvol:       maximum size of elemental tetrahedra
%plot (optional): type of plot to output (if any)
%              'element' if volume element plot is
required
%              'surface' if volume surface plot is
required
%              if no plot is required, omit input
argument
if nargin==3
```

```

        invert='no';
        plot='no';
        rotate='no';
    end

    format longG
    [I,II,III]=Volume;
    load;
    end

    [node,elem,face]=v2m(III,iso,opt,maxvol);
    node=0.01*node;
    if strcmp(plot,'element')
        plotmesh(node,elem)
    elseif strcmp(plot,'surface')
        plotmesh(node,face)
    end

    mat2inp('nodes.txt',node,'elements.txt',elem);
    end

```

Volume.m

```

function [I,II,III]=Volume(nimages,nmax)
clc;
if nargin<1
    nimages = 1016;
    nmax = 1016;
end
nmin = nmax-nimages+1;
II = uint8(zeros(875,1028,nimages));
j = 1;
for i = nmax:-1:nmin
    if (0 < i) && (i <= 9)
        prefix = '1__rec0005.000';
    end
    if (9 < i) && (i <= 99)
        prefix = '1__rec0005.00';
    end
    if (99 < i) && (i <= 999)
        prefix = '1__rec0005.0';
    end
    if (999 < i) && (i <= 9999)

```



```

        prefix = '1__rec0005.';
    end
    imagenumber = int2str(i);
    filename = strcat(prefix,imagenumber,'.tif');
    I = imread(filename);
    II(:,:,j) = I(:,:,1);
    j = j+1;
end
III=II(280:600,400:620,:);
save;
end

```

mat2inp

```

function
[NumNode,NumElem]=mat2inp(Nfilename,nodeM,Efilename,
elemM,Nrows,Erows)
%Function
%mat2inp: used to extract convert the 'node' and 'elem'
arrays into comma seperated text files
%
%Input Arguments
%Nfilename: filename to which node information will be
saved
%nodeM:      node array from which node text file is to be
written
%Efilename: filename to which element information will be
saved
%elemM:      element array from which element text file is
to be written
%Nrows (optional): number of nodes in mesh
%Erows (optional): number of elements in mesh

format longG
if nargin==4;
    [Nrows,Ncols]=size(nodeM);
    [Erows,Ncols]=size(elemM);
end
elemM(:,5)=[ ];
n=[1:Nrows]';
e=[1:Erows]';
NumNode=[n nodeM];
NumElem=[e elemM];
dlmwrite(Nfilename,NumNode,'delimiter',' ','precision',7);
dlmwrite(Efilename,NumElem,'delimiter',' ','precision',7);

```

end

porosity

```
function
[alpha]=porosity(SpecimenMatrix,rStart,rEnd,cStart,cEnd,zStart,zEnd)
%Function
%porosity: used to determine the porosity value of 3D
images based on
%           grayscale intensity value

%Input Arguments
%SpecimenMatrix: Input 3D Matrix (resembling specimen)
%rStart:      Beginning row for bounding box
%rEnd:        Ending row for bounding box
%cStart:      Beginning column for bounding box
%cEnd:        Ending column for bounding box
%zStart:      Beginning slice for bounding box
%zEnd:        Ending slice for bounding box

[r,c,z]=size(SpecimenMatrix);
if nargin==1
    rStart=1;
    rEnd=r;
    cStart=1;
    cEnd=c;
    zStart=1;
    zEnd=z;
end
totVol=(rEnd-rStart+1)*(cEnd-cStart+1)*(zEnd-zStart+1)
counter=0;
for i=zStart:1:zEnd
    for j=cStart:1:cEnd
        for k=rStart:1:rEnd
            if SpecimenMatrix(k,j,i)<=0
                counter=counter+1;
            end
        end
    end
end
counter
alpha=counter/totVol;
end
```