

Finite Element Analysis of Femur Bone Model

Gopi Jagini

A Thesis Submitted to
Indian Institute of Technology Hyderabad
In Partial Fulfillment of the Requirements for
The Degree of Master of Technology

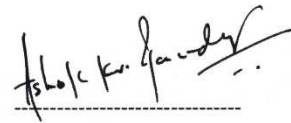


Department of Biomedical Engineering

July 2019

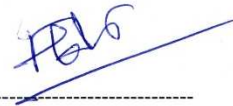
Approval Sheet

This thesis entitled Finite Element Analysis of Femur Bone Model by Mr. Gopi Jagini is approved for the degree of Master of Technology from Indian Institute of Technology, Hyderabad.



Dr. Ashok Kumar Pandey

Examiner
Mechanical and Aerospace Engineering
IIT Hyderabad



Dr. Falguni Pati

Examiner
Biomedical Engineering
IIT Hyderabad



Dr. Harikrishnan Narayanan Unni

Adviser
Biomedical Engineering
IIT Hyderabad

Declaration

I declare that this written submission represents my ideas in my own words, the ideas or words of others have been included, and I have adequately cited and referenced the original sources. I also declare that I have adhered to all principles of academic honesty and integrity and have not misrepresented or fabricated or falsified any idea/data/fact/source in my submission. I understand that any violation of the above will be a cause for disciplinary action by the Institute and can also evoke penal action from the sources that have thus not been properly cited, or from whom proper permission has not been taken when needed.

gopi

(Signature)

GOPI JAGINI

(Student Name)

BMITMTECH11002

(Roll No)

Acknowledgements

I, sincerely, express my gratitude towards my adviser Dr. Harikrishnan Narayanan Unni, Assistant Professor, Dept of Biomedical Engineering, IIT Hyderabad for his guidance, support and valuable inputs.

I would like to thank Ms. Tanu Khanuja, Research scholar, Biomedical Engineering, IIT Hyderabad for helping me in all stages of my work.

I would also like to thank Dr. Damodhar, Radiology Dept, MNR medical college and hospital, for providing me with CT image data required for my study.

I would like to take this opportunity to thank everyone who motivated and supported me throughout this project.

Abstract

Finite Element Method (FEM) is a numerical technique of obtaining solutions to boundary value problems. The practical application of Finite Element Method is called as Finite Element Analysis (FEA). FEA is the computational tool for performing engineering analysis. Biomechanical analysis involves dealing with the mechanical aspects of the biological systems. FEM techniques can be used to study the engineering analysis of different biological systems.

Femur bone is the longest and strongest bone of the human body. It undergoes mostly compressive loading. It exhibits anisotropy, indicating that the strength is different in different directions when a force is applied. It is hyperelastic in nature. It has two anatomical structures namely cortical bone tissue and cancellous bone tissue. The cortical is dense and tightly packed. The cancellous is porous. Cortical section is superior in mechanical properties compared to cancellous section. Cortical and cancellous are formed with collagen fibers. Upon the application of loading they align in the direction of loading, make bone stiff and gives strength to withstand a range of loads.

Femoral neck fracture is the common problem that occur in femur bone. This is more prominent in the patients with osteoporotic conditions. Osteoporosis is the condition in which the bone become highly porous and brittle. In the neck region, the amount of cancellous bone is more which makes it weaker. Under some high amount of forces, it creates a bending couple in the neck region and generated high stress regions. In this study, an attempt has been made to analyze the mechanical behavior of Femur bone model. A 3D realistic human femur bone has been modeled and finite element analysis has been performed using ABAQUS with different material models.

Contents

Approval Sheet	1
Declaration	2
Acknowledgements	3
Abstract	4
1. Introduction	9
1.1 Biomechanics	9
1.2 Finite Element Analysis	10
1.3 Femur Bone	11
1.4 Osteoporosis	12
2. Literature Review	13
3. Geometrical Modelling	17
4. Loading and Boundary Conditions	21
4.1 Loading and Boundary Condition	21
5. Finite Element Analysis: Different material models	23
5.1.1 Isotropic Linear Elasticity	23
5.1.2 Orthotropic Linear Elasticity	25
5.2 Mesh Optimization	26
5.3.1 Simulation results using ABAQUS: Linear Isotropic	28
5.3.2 Simulation results using ABAQUS: Orthotropic	30
6. Results and Discussion	32

7. Future Study	34
7.1 Osteoporosis	34
7.2 Implant Material Model	35
8. References	36

List of Figures:

1.1 Anatomy of Femur bone	11
1.2 Sectional view of cortical bone	12
1.2 Sectional view of cancellous bone	13
3.1 MRI and CT images of Femur bone	17
3.2 Generated 3D femur bone model	18
3.3 3D meshed model of femur bone model	19
4.1 Loading and Boundary condition	21
5.1 An Orthotropic material	25
5.2 Stress distribution for different meshed models	26
3.3 Mesh Optimization curve	27
5.4 Stress distribution in Cortical section (linear isotropic)	29
5.5 Stress distribution in Cancellous section (linear isotropic)	29
5.6 Stress distribution in Cortical section (orthotropic)	31
5.7 Stress distribution in Cancellous section (orthotropic)	31
7.1 Osteoporosis	34
7.2 Implant model	35

List of Tables

Table 5.1 Material properties for linear isotropic elastic analysis	28
Table 5.2 Orthotropic constants of cortical and cancellous sections	30

Chapter 1

Introduction

1.1 Biomechanics

Biomechanics is the branch of science which deals with the structure, function and motion of the mechanical aspects of biological systems. This study can be done at any level from whole organisms to organs, cells and cell organelles by using the methods of mechanics. The mechanical analysis of the biomaterials is carried out with the concepts of continuum mechanics. This approach is valid for the length scales above micro units. This continuum concept differs with the arguments when the length scales approach the order of micro, or even below.

Biomaterials are categorized into two groups: Hard and soft tissues. The mechanical analysis of hard tissues (like bone) can be analyzed with the help of linear/non-linear elasticity theories. Whereas, soft tissues (like Skin and cartilage) undergo huge amount of non-linear deformations. The computational analysis of soft tissues can be carried out using finite strain theory and computer simulations.

Computational biomechanics is the application of engineering computational tools, such as the Finite element method to study the mechanics of biological systems. These Computational models and simulations can be used to predict the relationship between different parameters. Different computational models can be used to design more relevant experiments, thereby reducing the time and costs of experiments. Mechanical modeling using finite element analysis has been the hot topic for a few years now, used to interpret the experimental observation of the behavior of different biological systems. In medicine, over the past few years, the finite element method has become a great alternative to in vivo surgical assessment. One of the great advantages of computational biomechanics is, its ability to determine the endo-anatomical response of an anatomical structure, without being subject to ethical restrictions [1].

The study of biomechanics ranges from the inner workings of a cell to the movement and development of limbs. It includes finding the mechanical properties of soft tissues and bones. The biomechanical research can be used to investigate the forces on musculoskeletal system, such as

different forces on limbs. Biomechanics has its wide usage in orthopedic applications, to design implants for different human joints, dental parts, external fixations and medical support systems.

1.2 Finite Element Analysis

Finite element method (FEM) is a technique of solution of the boundary value problems [2]. It is a numerical method of obtaining solutions of differential and integral equations. Finite Element Analysis (FEA) is the practical application of FEM. FEA is the computation tool for investigating the engineering analysis. FEA is used for analyzing newly designed products as well as for already existing designs using the equations of mechanics of material [3].

The application areas of Finite Element Method (FEM) include structural analysis, heat transfer, fluid flow, mass transport and electromagnetic potential, etc. To solve a problem, FEM subdivides the whole system into smaller, simpler parts which are called finite elements. It then uses variational methods from calculus to approximate the solution by minimizing an associated error function. The subdivision of whole domain into smaller parts helps in accurate representation of a complex geometry. It will also help in providing dissimilar material properties to different regions. It has the advantage of capturing the local affects.

In structural analysis, the simulations help us in providing stiffness and strength visualizations. FEM allows detailed visualizations of stress and displacement distributions. FEM packages provide different variety of options to control the complexity, simulation methods and visualization techniques. A product, in FEM, can be constructed, refined and optimized before its actual manufacturing.

FEA is a good choice for analysis over complicated domains. It includes the use of mesh generation techniques for dividing a complex problem into smaller, simpler elements. The mesh is an integral part of FEA and it must be controlled carefully to obtain accurate results. Generally with the increase in number of mesh elements, the accuracy of solution increases for a discretized problem. However after a range of increment, the results converge and further mesh refinement will not increase the accuracy of results [4]. FEA also uses software program coded with FEM algorithm. FEA simulations provide a valuable resource as they remove multiple instances of creation and testing of hard prototypes.

1.3 Femur Bone

The Femur bone, also known as Thigh bone, is located within the human thigh. It is both the longest and strongest bone in the human body, extending from the hip to the knee. The length of this bone is almost 26% of height of the person. Important features of this bone include the head, medial and lateral condyles, patellar surface, medial and lateral epicondyles, and greater and lesser trochanters. The head is where the bone forms the hip joint with the innominate bone. The condyles are the points of connection with the tibia, which is a lower leg bone. The patellar surface is the groove where the bone adjoins with the patella, or knee cap. The epicondyles and trochanters are all important attachment sites for various muscles. Bone matrix consists of collagen fibers and organic ground substance, primarily hydroxyapatite formed from calcium salts.

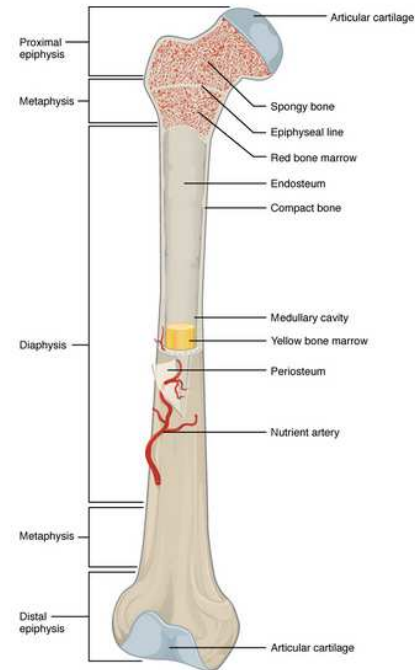


Fig 1.1. Anatomy of femur bone

For structural analysis, femur bone can be assumed as an assembly of two different sections. One is the Cortical or Compact bone and other is the Cancellous or Spongy bone. Cortical bone is dense and composed of osteons. Cortical bone can withstand compressive forces. While the Cancellous bone is porous, less dense and composed of trabeculae. It supports shifts in weight distribution. It is mainly responsible for energy absorption. Cortical bone is strong compared to cancellous bone.

The shaft region of femur bone is almost cylindrical in shape. It contains more cortical bone compared to cancellous bone. The neck and ball regions of femur are dominated by the presence of cancellous bone. Upon loading the collagen fibers align along the length of the shaft and make the bone stiffer. When the load is released, the fibers will be relaxed.

The hierarchical structure of bone significantly contributes to high stiffness, strength, toughness and energy absorption, light weight and other remarkable mechanical properties of bone. The bone is composite and inhomogeneous. It exhibits anisotropy and it shows hyperelastic behavior. The anisotropic behavior of bone is due to the distribution of collagen fibers

and their orientation [5]. The mechanical properties are direction dependent. Femur bone is mostly under compression and the properties are superior in the direction of longitudinal axis.

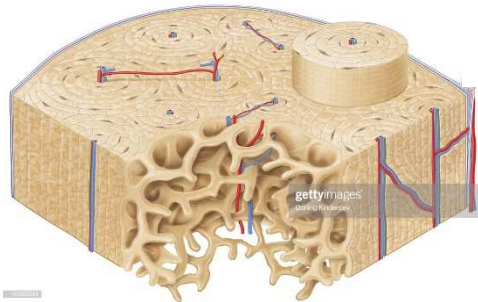


Fig 1.2. Sectional view of cortical bone

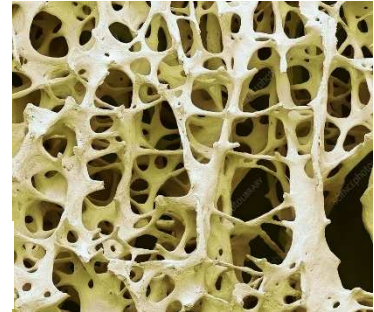


Fig 1.3. Sectional view of cancellous bone

The human femur can withstand forces of 1,800 to even 2,500 pounds, so it is not easily fractured. Only a large amount of force can generate crack or break in femur bone, such as a car accident or a fall from an extreme height. Such an injury can take a few months to heal.

1.4 Osteoporosis:

Osteoporosis is a condition which makes the bone weak and brittle. In this condition, a fall or even mild stress can fracture the bone. The most common fracture of osteoporotic condition is the femoral neck fracture.

Bone tissue is constantly being broken down and regenerates on its own. But when the regeneration rate is lesser than the deterioration rate, the bone becomes highly porous and leads to the condition of osteoporosis. It is mostly seen in old aged people. It effects men and women of all races. But women are at higher risk.

Femoral fracture is one type of hip fracture, occurs in the neck region below the ball portion of femur neck. In the neck region the amount of cancellous bone is more and with the addition of osteoporosis, it becomes highly porous and fragile. In this, the ball is disconnected form the thigh bone. In order to restore its functionality, partial or total hip replacement surgery will be done. In total hip replacement surgery, the ball portion will be removed and an artificial stem-ball-socket arrangement will be installed.

Chapter 2

Literature Review

In this chapter, a brief description is provided on the past work in the field of Biomechanical analysis of bone model using finite element analysis techniques and the work related to generating the material properties.

The bone tissue and strong and rigid structure of human body. It behaves so due to its composition, the combination of organic and inorganic materials, their distribution and their arrangement. The minerals calcium, phosphate along with collagen forms the organic part which make 60-70% of bone tissue. Bone tissue is anisotropic in nature, indicating that the bone behavior will change according to the direction of the applied load. Femur bone is strong to withstand loads in longitudinal direction [6].

The skeletal system undergoes different types of loads. The bone receives loads in different directions. Bone undergo all different types of loads will be applied: compressive, tensile, shear or torsion. The compressive strength of hip joint decides the danger of femoral neck injury. The hip joint absorbs three to seven times body weight during normal walking. It might go up, fifteen to twenty times in case of jumping. In normal standing position, with both legs on ground, the hip joint takes about one third of body weight.

In standing or in the stance position of walking or may be running, the type or load will be bending on femoral neck. It creates high compressive stress on the lower neck region and high tensile stress on the upper neck region. The bones fracture easily under shear loading, due to its anisotropic nature. The bone can also develop a fracture if the generated compressive or tensile stress values go beyond the strength values. A low amount of force with high frequency can also fracture the bone tissue easily [6].

Marek Pawlikowski et al presented a study of a new consecutive model for defining human trabecular bone [7]. It takes into account the non-linear viscoelastic behavior of human cancellous bone tissue. The elastic behavior of the tissue is characterized by the hyperelastic Mooney-Rivlin model and the viscoelastic behavior is characterized by means of a hereditary integral, which

describes the dependency of stress on both strain and time. The required material properties are obtained from the stress relaxation tests and indentation tests using suitable curve fitting techniques. The Mooney-Rivlin properties, C10 and C01 for trabecular bone samples of four different age groups are calculated. The viscoelastic properties are also calculated.

Morphologically, bone exhibits two types of anatomical structures, i.e. cortical (compact) bone and cancellous (trabecular) bone. Osteon is the basic unit of cortical bone and trabecula is the basic unit of cancellous bone. Both, Osteons and trabecule consist of collagen fibers reinforced with calcium phosphate particles. In macro scale bone can be considered as a composite formed from osteons, trabecule and body fluids [7].

Uzair N. Mughal et al presented a study on FEM analysis of femur bone [2]. They analyzed the stresses experienced by the femur bone. They developed a 3D CAD model of generic human femur bone model. They have considered linear isotropic modelling of femur bone and analyzed the stress distribution.

3D laser scanning is performed to create the femur bone model. The data is imported to SolidWorks and using the techniques of transforming 2D into 3D models, the structure is generated. However the marrow section is approximated to a hollow cylindrical structure of radius of 1.6cm. The whole femur bone is considered as a single unit with a hollow cylindrical cavity in it. Tetrahedral 20 node type elemental mesh is created and the simulations are carried out using ANSYS.

Linear static analysis is being carried out ignoring the role of muscles in sharing the load. A fixed boundary constraint is applied on the lower end of the bone. Two different kinds loading conditions are applied. In first case axial compressive load is applied to correlate the conditions of weight handled by the femur in standing position. In second case bending load is applied and the stress distribution was studied [2].

K. V. Arun et al presented their work on the behavior of femur under bending and impact loads [8]. Femoral fractures are the most common injuries. The biomechanical analysis of bone plays an important role in the evaluation and treatment of femoral fractures, thereby helping the surgeons to treat the patients effectively. Impact resistance, fracture toughness and bending strength are the main characteristics that influence the fracture damage of the bone.

Experimental investigations are carried out to investigate above mentioned properties. The results show that the femur strength is highly variable in various regions of the bone. Above each parameter has maximum dependency on bone matrix and its hardness [8].

Noritoshi ATSUMI et al presented a constitutive model of cortical bone considering anisotropic inelasticity [9]. The damage evaluation is developed to predict possible injuries accurately. For cortical bone, the anisotropy of mechanical properties like elastic modulus and yield stress is expressed with strain rate dependency and asymmetric stress vs strain curves for both compression and tension.

The strength of cortical bone decreases in the following fashion: Compression, Tension and Torsional loading. The mechanical properties of cortical bone depend on the direction of loading, strain rate difference and loading pattern (compression or tension). In case of tension, the bone fractures in transverse direction where as in compression fracture occurs diagonally with respect to the axis. The type of fracture in case of torsional loading is spiral [9].

A.E. Yousif et al presented their study on biomechanical analysis of femur bone during normal walking and standing up [10]. The loads on femur from hip joint during dynamic activities have been studied. The direction of force vector on femur head is decided by the external forces acting on the limb and the internal forces generated by muscle contraction.

3D femur bone model is reconstructed from CT (DICOM) images. Whole model is designed as two sections, outer cortical and inner cancellous bone. The analysis is performed on ANSYS work bench. Both the sections were meshed in ANSYS and are given different material properties. Both the cortical and cancellous sections are considered linear orthotropic and the properties are assigned accordingly. The stress distribution is studied showing high stress regions in the neck region of femur [10].

Majid Shahzad et al presented a study on mechanical characterization and Finite element modelling of a hyperelastic material [11]. They characterized the hyperelastic material properties with a suitable strain energy function for an indigenously developed rubber material.

Different tests like uniaxial, volumetric tests along with equi-biaxial and planar shear tests are performed and data is collected. With the help of Digital Image Correlation (DIC) they calculated strain measurements to give input stress vs strain data in ABAQUS.

In the property module, the uniaxial, biaxial and volumetric data is given as input and with suitable curve fitting the material coefficients for different hyperelastic strain energy functions are calibrated. These material coefficients exactly define the hyperelastic behavior of the developed rubber material and can be used in different loading simulations.

Chapter 3

Geometrical Modelling

The first stage of this study deals with the generation of a 3D realistic bone model. In the next stage, the model has been imported to ABAQUS 6.14[©] for simulation and the analysis has been carried out. For generated 3D bone model, CT scanned images of femur bone have been used. The images were collected from Radiology department, MNR medical college and hospital. The thigh bone of a 26 years male was scanned and the data was collected in DICOM images.

In order to create the model, the DICOM images were converted to high quality JPEG images and were imported to AMIRA 5.6[©] software. The model can be created with the help of either MRI or CT scanned images. The problem associated with MRI images is that different sections of bone are not clearly identifiable. It was hard to differentiate between cortical and cancellous sections of femur bone. Whereas the CT scanned images are clearly showing different sections of bone, cortical and cancellous regions are well differentiated.

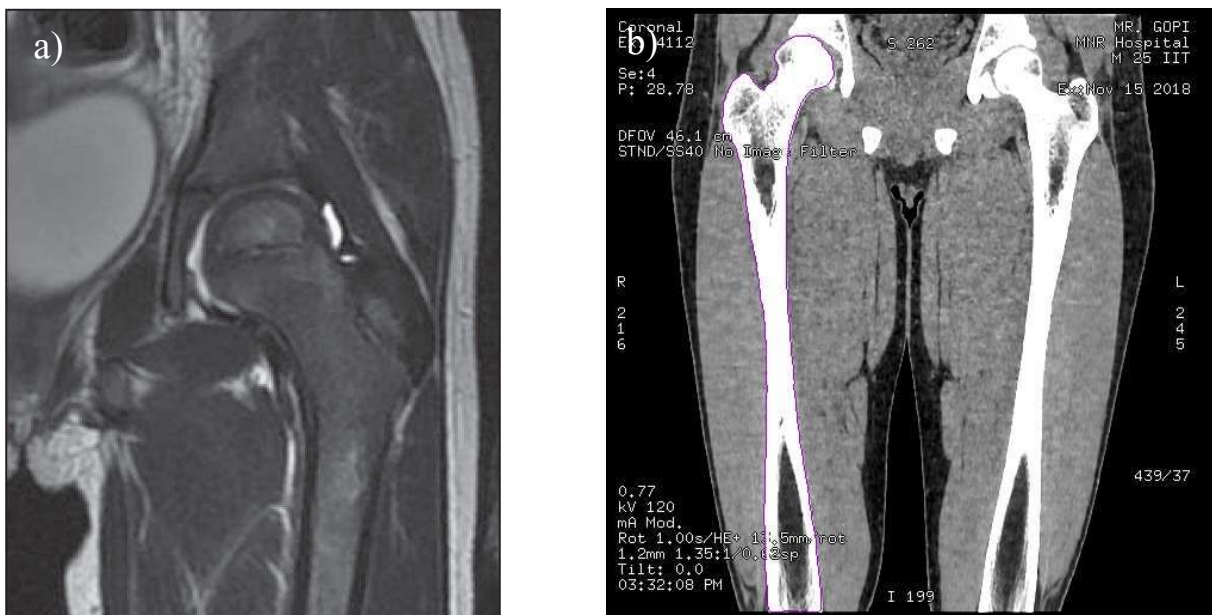


Fig 3.1. a) MRI image of femur bone. b) CT scanned image of femur bone (with clear identification of cortical and cancellous sections).

The collected CT images, 83 sectional images, were loaded in AMIRA and segmentation tools were used to generate the model. A boundary box has been created with length, width and thickness as 0.4, 0.4 and 0.06 respectively in order to match the model dimensions with the actual femur bone. The volume rendering has been done to check the bone shape, so that we can make changes in bounding box to make it look correct in shape.

In the segmentation tool, a new label has been created and two materials were named to assign the segmented regions accordingly. One for the cortical segmentation and the other for the cancellous section. From each slice/image both the cortical and cancellous sections were selected manually and were assigned accordingly to the above created materials. After completion of selecting respected areas of both the parts, the 'Surface Gen' module was applied on the previously created label. With the help of 'Surface View' module both the cortical and cancellous surface can be viewed as shown below.

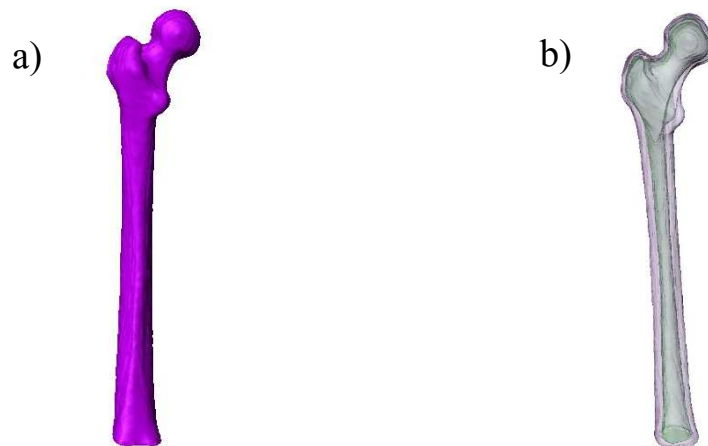


Fig 3.2. a) Generated 3D model of Femur bone. b) Transparent model showing cortical (outer) and cancellous sections (inner)

In the label module, with different available options, the surface can be made smooth. In the 'Surface Editor' module by specifying the distance between the nodes, the surface can be refined. For e.g. by specifying the distance between the nodes as 0.05, the generated surface has around 18500 triangular faces and by specifying the distance between the nodes as 0.1, the newly generated surface has around 3600 triangular faces in it. With the increase in value of the distance

between the nodes, the surface will be generated with lesser and bigger triangles which makes the surface look a bit rough. With more number of triangular surfaces, the surface looks smooth and close to the true shape of the actual bone. In order to carry out the mesh optimization for analysis in ABAQUS, different models with different number of surfaces have been generated.

The biggest advantage of using AMIRA for geometrical model generation is that the surface model can be converted to solid model by generating tetrahedral elements within the defined boundaries of the surfaces. Later the model can be directly imported to ABAQUS for analysis. This can be done by saving the tetra-generated file into '.inp' format and importing it directly as a model to the ABAQUS.

In the label by applying 'Tetra Gen' module the surface model can be converted to solid model with tetrahedral elements. For the model with 18500 faces, it has generated around 79000 elements and for the model with 8200 faces it has generated the solid model with 27000 elements. The two surfaces will generate the solid sections with the nodes at the touching boundaries as common nodes for both the sections. Figure 3.3 below shows the generated tetrahedral elemental model of the femur bone.



Fig 3.3. The 3D meshed model of Femur bone model with tetrahedral elements

The default type of element, in tetra generation, in AMIRA is a 4-node tetrahedral element. The generated meshed model will be imported as an orphan mesh model to the ABAQUS. In ABAQUS the orphan mesh model can be edited and the element type can be modified to 10-node tetrahedral for analysis.

Before generating the 'Tetra Gen', the surface model should be assessed to meet the below mentioned criteria. These help us check the quality of the model before generating the model. The quality check will be done in the label module. A decent model should

- 1) have no intersections
- 2) have no triangles with bad orientation
- 3) have aspect ratio below 20 or less
- 4) have dihedral angle 10 or above
- 5) have tetra quality below 25 or less
- 6) have no holes

If the surface model violates any of the above mentioned points then flip the edges of the wrong triangles until it meets the above criteria. This repair can be done automatically or manually in AMIRA. After the quality check the 'Tetra Gen' can be applied and the solid model can be generated. From the '.grid' module, after tetra gen, the model can be saved to '.inp' file format which can be imported directly to the ABAQUS.

Chapter 4

Loading and Boundary Conditions

The earlier chapter described the geometrical modelling of 3D bone model. This chapter discusses a few things related to loading and boundary conditions. Once the model was ready, it was time to decide the nature of loading and its location, defining boundary conditions and optimizing the number of mesh elements of the model before going for simulation.

4.1. Loading and Boundary Conditions

Most of the time the Femur bone undergoes compressive loading. So, the nature of loading has been decided to be compressive impact loading in the direction, assumed longitudinally along the shaft of the Femur. The load is applied on the ball portion of the femur model, which represents the loading condition from hip joint to the femur bone.

A fixed boundary condition, restricting all the translational and rotational movements, at the base of the shaft is assumed. The actual knee joint allows rotational movement of the femur bone in dynamic conditions. But in case of standing, in a fixed stature, the movements are constrained forcefully to make it look like a fixed joint at the base of the femur. Figure below shows the loading and boundary conditions on the femur model for simulation.

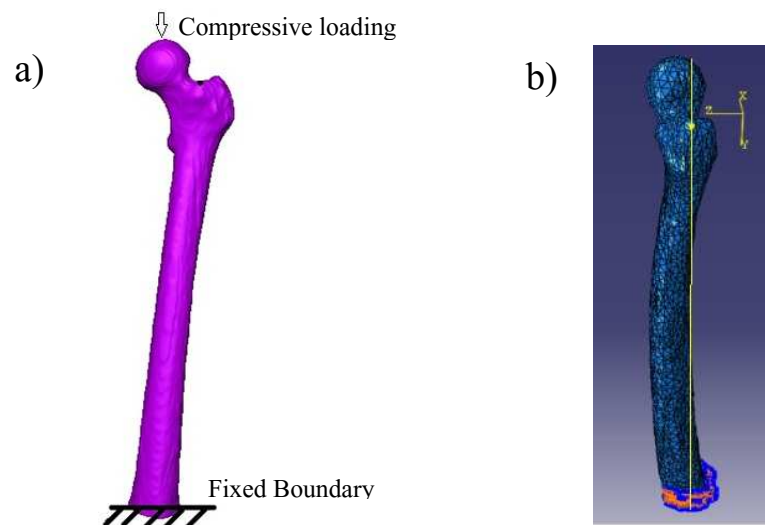


Fig 4.1. a) Loading and Boundary Condition. b) Longitudinal axis of the model

During normal walking of a human being in the gait cycle, the maximum compressive load applied on the femur bone is 1700N for a 75kg male. This force will be less when a person is in standing position. Here for our study, the max load of the gait cycle in walking, 1700N was taken and it was applied in the assumed longitudinal direction. The loading type is compressive. All other muscle forces and any other forces in other two lateral directions were neglected due to their minute values compared the load in longitudinal direction.

Chapter 5

Finite Element Analysis: Different Material Models

In the earlier chapters, modelling of the human femur bone, loading type, boundary conditions and mesh optimization have been discussed. This chapter explains the analytical results on different material models, like Linear Isotropic, Orthotropic Linear Elasticity and Hyperelastic model. The model with mesh elements has been imported and analysis has been carried out in ABAQUS 6.14[©]

To carry out the mechanical analysis, the meshed model with 79000 tetrahedral elements (generated in AMIRA) was used. The default 4-node tetrahedral elements were modified to 10-node tetrahedral elements in ABAQUS mesh module for accurate results. The model was imported as an assembly of two sections with a surface to surface contact.

5.1.1 Isotropic Linear Elasticity

Bone is hyperelastic and anisotropic in nature, indicates that the mechanical properties are dependent on orientation. This section deals with the mechanical analysis of bone by assuming it as a linear isotropic elastic material. Both the cortical and cancellous sections are assumed to be linear elastic isotropic.

The general representation of the Hooke's law for a linear elastic material is shown below. Where each stress component σ_i depends linearly on all strain components ε_j .

Stress as a function of strain: $\sigma = CE$.

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \sigma_4 \\ \sigma_5 \\ \sigma_6 \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & C_{14} & C_{15} & C_{16} \\ C_{21} & C_{22} & C_{23} & C_{24} & C_{25} & C_{26} \\ C_{31} & C_{32} & C_{33} & C_{34} & C_{35} & C_{36} \\ C_{41} & C_{42} & C_{43} & C_{44} & C_{45} & C_{46} \\ C_{51} & C_{52} & C_{53} & C_{54} & C_{55} & C_{56} \\ C_{61} & C_{62} & C_{63} & C_{64} & C_{65} & C_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \varepsilon_4 \\ \varepsilon_5 \\ \varepsilon_6 \end{bmatrix} \quad (5.1)$$

The 36 C_{ij} notations are material constants called stiffnesses and are calculated from experiments. The above 6x6 matrix is called the **stiffness matrix (C)**. The strain energy formulation in elasticity theory shows that it is necessary for the stiffness matrix to be symmetric and so there are only 21 independent elastic constants in the most general case, i.e., anisotropic elasticity. By inverting the above equation 5.1.1, the strain components can be expressed as a function of stress components: $E = S\sigma$. Where S is termed **compliance matrix**.

To define Isotropic Linear Elasticity, the symmetry of stiffness matrix is reduced to two independent elastic constants. The material response is independent of orientation in case of isotropic materials. The stiffness matrix is defined as:

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \sigma_4 \\ \sigma_5 \\ \sigma_6 \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{12} & 0 & 0 & 0 \\ 0 & C_{11} & C_{12} & 0 & 0 & 0 \\ 0 & 0 & C_{11} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{11} - C_{12} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{11} - C_{12} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{11} - C_{12} \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \varepsilon_4 \\ \varepsilon_5 \\ \varepsilon_6 \end{bmatrix} \quad (5.2)$$

The stress components can be expressed in terms of **Lame Constants**, λ and μ .

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \sigma_4 \\ \sigma_5 \\ \sigma_6 \end{bmatrix} = \begin{bmatrix} \lambda + 2\mu & \lambda & \lambda & 0 & 0 & 0 \\ 0 & \lambda + 2\mu & \lambda & 0 & 0 & 0 \\ 0 & 0 & \lambda + 2\mu & 0 & 0 & 0 \\ 0 & 0 & 0 & 2\mu & 0 & 0 \\ 0 & 0 & 0 & 0 & 2\mu & 0 \\ 0 & 0 & 0 & 0 & 0 & 2\mu \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \varepsilon_4 \\ \varepsilon_5 \\ \varepsilon_6 \end{bmatrix} \quad (5.3)$$

$$\text{Where, } \lambda = \frac{E\nu}{(1+\nu)(1-2\nu)} \text{ and } \mu = \frac{E}{2(1+\nu)} = G \quad (5.4)$$

E , ν and G are the engineering constants called Modulus of Elasticity, Poisson's ratio and Modulus of Rigidity respectively.

5.1.2 Orthotropic Linear Elasticity

The nature of Orthotropic material is, it has three orthogonal planes of microstructural symmetry. A representative illustration of orthotropic material is shown in the fig 5.1. The material is formed with thousands of very slender, long fibers bound together in bundles with oval cross-sections. These bundles are put together in place with the help of a binder material. The fine microstructural details of the bundles and surrounding matrix are smeared out and averaged. Principal direction will be along the longitudinal direction of fibers.

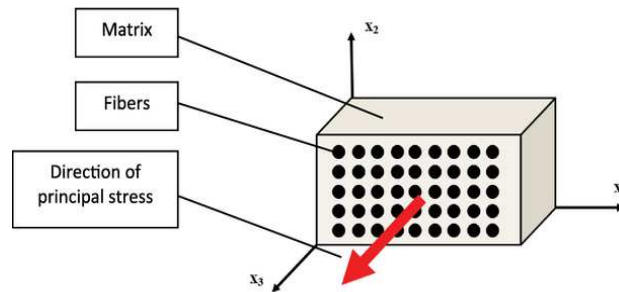


Fig 5.1 An orthotropic material

The material symmetry inherent in the orthotropic material further reduces the number of independent constants from 21 (in most general case) to **9**. From equation 5.1, the stiffness matrix is thus reduced to below form

$$\begin{bmatrix} \sigma_1 \\ \sigma_2 \\ \sigma_3 \\ \sigma_4 \\ \sigma_5 \\ \sigma_6 \end{bmatrix} = \begin{bmatrix} C_{11} & C_{12} & C_{13} & 0 & 0 & 0 \\ 0 & C_{22} & C_{23} & 0 & 0 & 0 \\ 0 & 0 & C_{33} & 0 & 0 & 0 \\ 0 & 0 & 0 & C_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & C_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & C_{66} \end{bmatrix} \begin{bmatrix} \varepsilon_1 \\ \varepsilon_2 \\ \varepsilon_3 \\ \varepsilon_4 \\ \varepsilon_5 \\ \varepsilon_6 \end{bmatrix} \quad (5.5)$$

The above equations can be inverted to get the compliance matrix in terms of engineering constants $E_1, E_2, E_3, \nu_{12}, \nu_{13}, \nu_{23}, G_{12}, G_{13}$ and G_{23} . Where E_i represents the Young' modulus (stiffness) in respective direction, ν_{ij} represents Poisson's ratio and G_{ij} (shear modulus) represents the shear stiffness in the corresponding plane.

5.2 Mesh Optimization

Mesh optimization involves defining the range of number of tetrahedral elements for meshing the volumetric model so that the simulation results are invariant with the number of tetrahedral elements. In order to carry out mesh optimization, different models of the bone with different number of elements were generated. Models with different number of elements can be generated either by specifying number of triangular surfaces or by providing the distance between the adjacent nodes (which automatically generates a specified number of surface triangles).

To calibrate optimum mesh density, four models with different number of elements were generated in AMIRA. First one with a minimum nodal distance of 0.005, 18800 faces and 79000 elements. Second one with a nodal distance of 0.006, 13600 faces and 51000 elements. Third one with a nodal distance of 0.007, 10300 faces and 36000 elements. The last one with a nodal distance of 0.008, 8200 faces and 27000 elements.

For all the above models, surface quality check was performed in AMIRA and then tetrahedral generation was applied. Later, each of these models were imported to ABAQUS and properties were assigned for cortical and cancellous parts separately (Orthotropic linear elastic material properties were assigned). The above mentioned loading type and boundary conditions were applied. A compressive force of 1700N was applied on each of the models, static impact analysis was carried out, and the maximum principal stress values were recorded for each case, as shown in Fig 4.2.

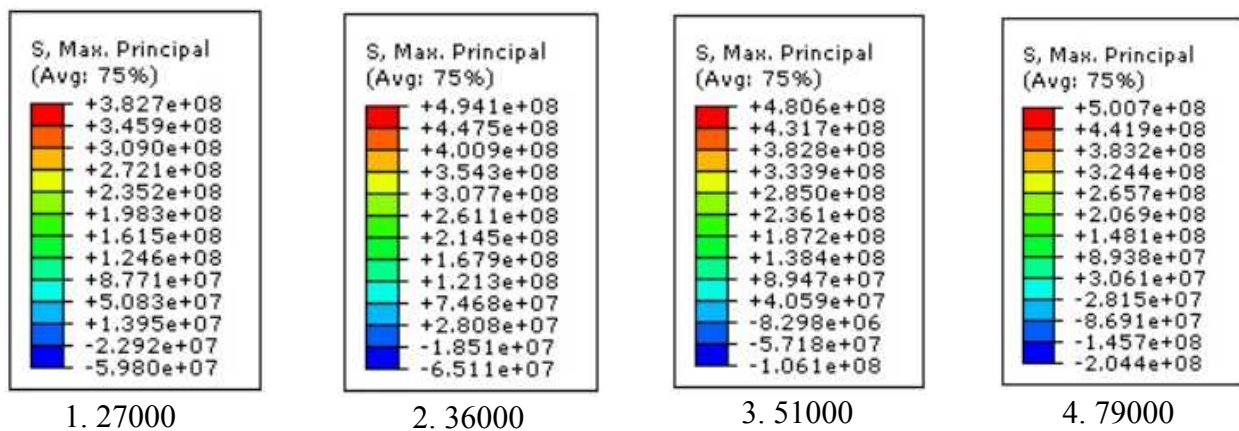


Fig 5.2. The maximum principal stress distribution for models with different number of elements

The maximum stress values in each case were plotted against number of mesh elements as shown in Fig 4.3. On the Y-axis the stress values were marked in MPa and on the X-axis the number of elements were marked in thousands.

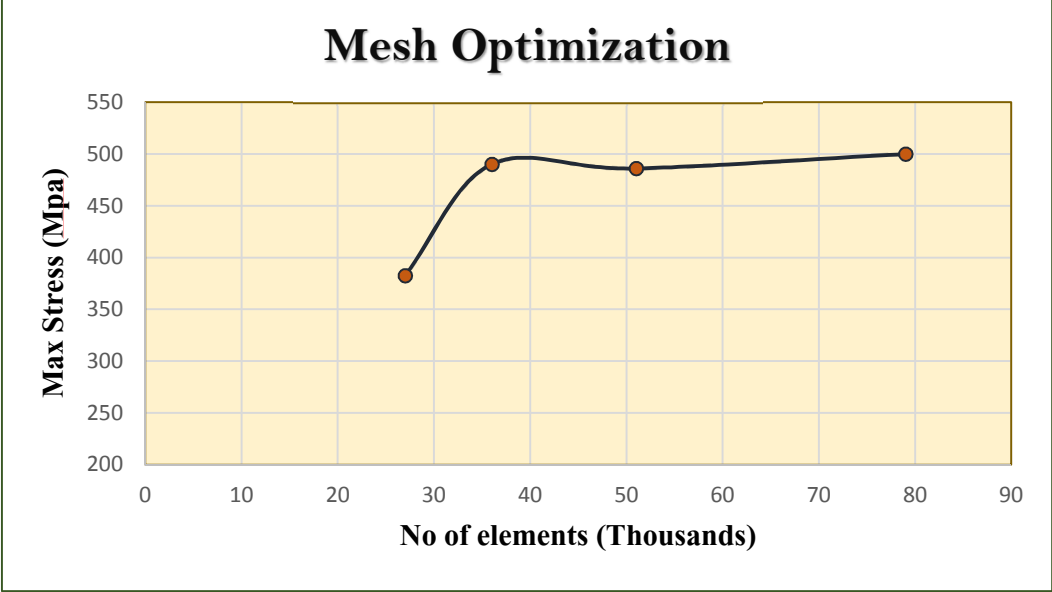


Fig 5.3. Graph showing the maximum stress vs number of mesh elements

It can be observed from the graph that after a value of 36000 number of mesh elements there was not much significant difference in the maximum stress values and they were invariant with number of mesh elements further. So, for mechanical analysis in ABAQUS, the femur bone model with 79000 mesh elements has been considered.

5.3.1 Simulation results using ABAQUS: Linear Isotropic

After defining the material model and the optimized mesh size, the next step was to analyze the model using ABAQUS. The meshed bone model has been imported to ABAQUS and the material properties, as shown in the below table 5.1, were defined in the property module for cortical and cancellous sections individually. Elastic Modulus (Pa), Density (kg/m³) and Poisson's ratio were given as the inputs. In the section module, the properties were assigned to respective regions of the model.

Section	Elastic Modulus, E(GPa)	Poisson's Ratio, μ	Density, $\rho(\text{kg/m}^3)$
Cortical	18	0.3	2000
Cancellous	3.2	0.23	1300

Table 5.1 The properties for linear isotropic elastic analysis [10]

The imported orphan mesh was modified to new 10-node element type, keeping same number of elements. In the load module, the compressive load was defined, along the Y-direction (assumed longitudinal axis) of the bone model, on the ball region and a magnitude of 1700N was given. In the boundary condition module, the bottom face of the model was given a fixed boundary (by restricting all the translational and rotational movements). The load was applied for a duration of 1 sec and the stress distribution was studied. The stress distribution is as shown below.

The maximum principal stress distribution for the cortical section is shown in below fig 5.4. The regions of highest stress are located in the neck and top portion of the shaft region. In the neck region the applied load is transverse to the loading direction which acts as a bending load and creates high compressive stress in the lower neck and high tensile stress in the upper neck region with a value of 0.465 GPa.

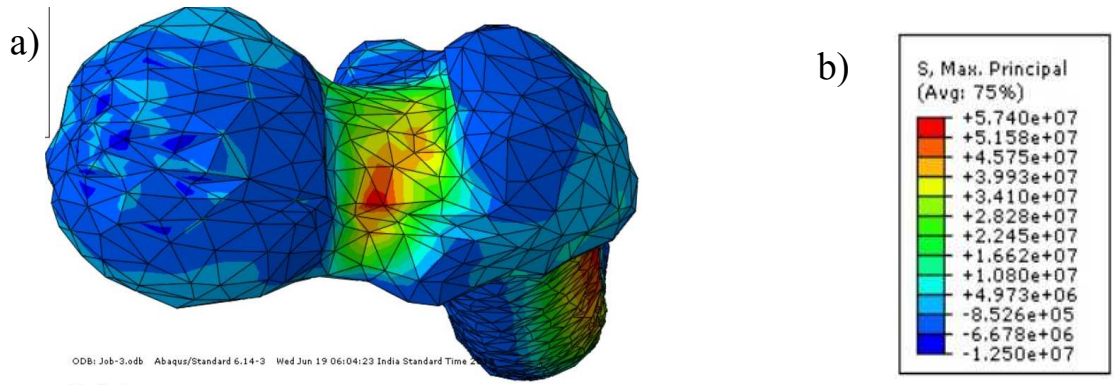


Fig 5.4 a) Stress distribution in cortical section. b) The maximum principal stress values

The maximum principal stress distribution for the cancellous section is shown in below fig 5.5. The regions of highest stress are located in the neck region of the cancellous bone. In the neck region the applied load is transverse to the loading direction, acts as a bending load, which creates high compressive stress in the lower neck and high tensile stress in the upper neck region with a value of 0.057 GPa.

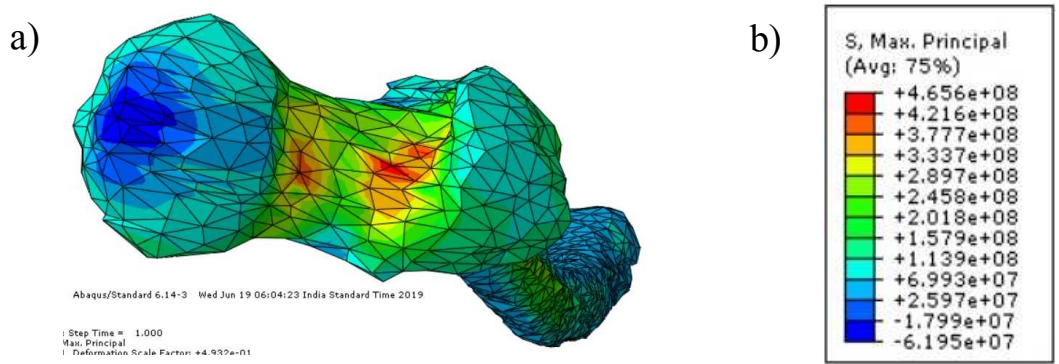


Fig 5.5 a) Stress distribution in cancellous section. b) The maximum principal stress values

5.3.2 Simulation results using ABAQUS: Orthotropic

For the mechanical analysis, the bone model was imported to ABAQUS and properties were assigned. The orthotropic properties shown in the below table 5.2.1 were assigned to the cortical and the cancellous sections respectively. Same loading and boundary conditions were applied, as discussed in the previous case, and the stress distribution was obtained.

Parameter	Cortical bone	Cancellous bone
Density (kg/m ³)	2000	1300
Modulus of Elasticity (Mpa)	$E_1 = 6982.9$ $E_2 = 6982.9$ $E_3 = 18155$	$E_1 = 2029.4$ $E_2 = 2029.4$ $E_3 = 3195.3$
Poisson's Ratio	$\nu_{12} = 0.4$ $\nu_{23} = 0.25$ $\nu_{13} = 0.25$	$\nu_{12} = 0.4$ $\nu_{23} = 0.25$ $\nu_{13} = 0.25$
Shear Modulus (Gpa)	$G_{12} = 4.69$ $G_{23} = 5.61$ $G_{13} = 7.68$	$G_{12} = 4.69$ $G_{23} = 5.61$ $G_{13} = 7.68$

Table 5.2 The orthotropic constants of cortical and cancellous sections [10].

Before carrying out the simulations, in ABAQUS, for any type of anisotropic material, it is required to define a local coordinate system. This helps in defining the principal direction. For this, direction “3” is assumed as the direction of principal stress, which is the Y-direction (The longitudinal axis of the bone).

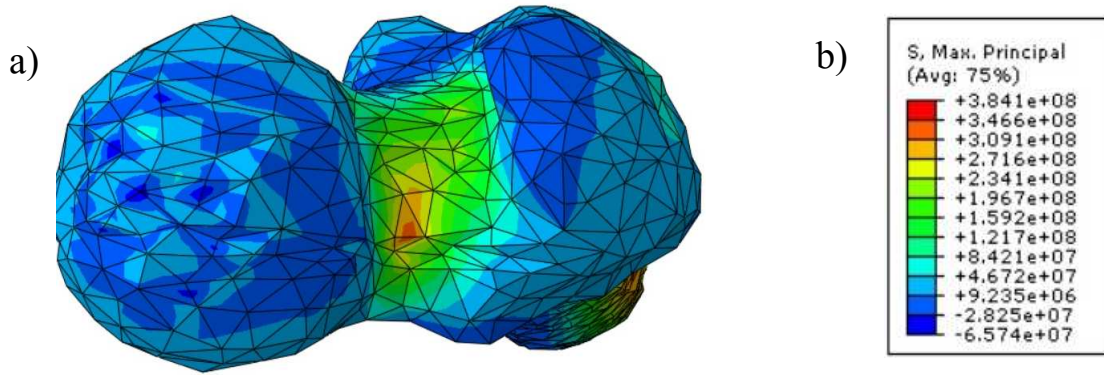


Fig 5.6 a) Stress distribution in cortical section. b) The maximum principal stress values

The maximum principal stress distribution for the cortical section is shown in above fig 5.6. The regions of highest stress are located in the neck and top portion of the shaft region. In the neck region the applied load is transverse to the loading direction which acts as a bending load and creates high compressive stress in the lower neck and high tensile stress in the upper neck region. The maximum stress value is 0.384 GPa.

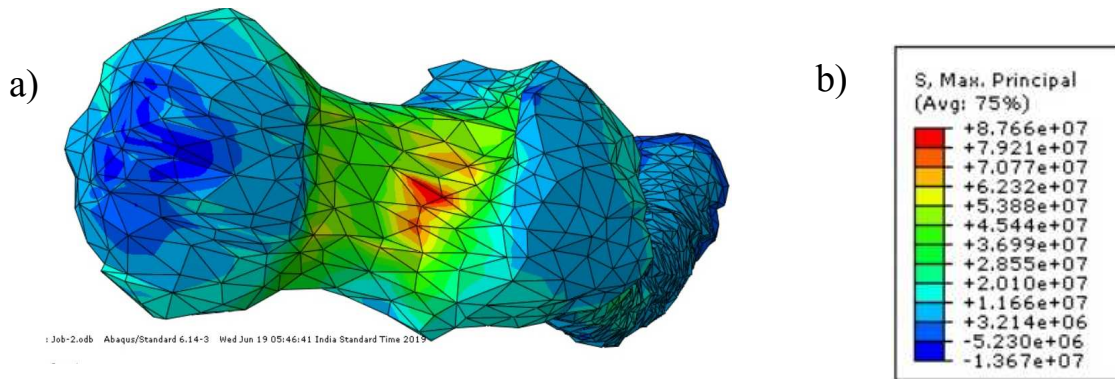


Fig 5.7 a) Stress distribution in cancellous section. b) The maximum principal stress values

The maximum principal stress distribution in the cancellous portion of the bone model is shown in above fig 5.7. The regions of highest stress are located in the neck region of the cancellous bone. In the neck region the applied load is transverse to the loading direction, acts as a bending load, which creates high compressive stress in the lower neck and high tensile stress in the upper neck region. The highest stress value is 0.087 GPa.

Chapter 6

Results and Discussion

Earlier chapters described the details of geometrical modelling, loading, boundary conditions, mesh optimization and the maximum principal stress distribution with different material models. This chapter discusses about the obtained results and conclusions that can be drawn from the results.

With the help of CT images, the 3D realistic human femur bone model has been created to imitate the geometrical features of the femur bone. The model was created as an assembly of two main sections of the bone, cortical and cancellous part. The model then meshed in AMIRA and quality assessment was performed before analysis. The meshed model, taken as an orphan mesh in ABAQUS, cannot be refined in ABAQUS in terms of mesh density. The element type can be changed from 4-node to 10-node tetrahedral without altering the number of mesh elements.

In first case, the Isotropic linear elastic behavior is assumed for both the sections of the bone with different material constants. A compressive load of 1700N is applied along the longitudinal direction of the bone, with bottom of the shaft as a fixed boundary, for a duration of 1 sec. 1700N load is the maximum applied load on femur by the hip joint in the gait cycle of a 75kg male human while walking. The average duration of gait cycle is around 1.1sec [10].

In the cortical section, the maximum stress of 0.465GPa, was appeared in the neck region. In the shaft portion also high stress values are obtained. This is because, in the neck region, even in shaft portion, the applied load would create a bending couple and that creates high compressive stresses in the lower neck and high tensile stresses in the upper neck region. The same phenomena happens in the cancellous section of the bone. The maximum stress value in cancellous part is 0.057 GPa which is lower compared to the values obtained in cortical section, because the cortical section is stiffer compared to cancellous section so it produces high amount of stresses compared that of cancellous stress.

In the second case, the orthotropic behavior for both the cortical and the cancellous sections and assumed. The properties were assigned accordingly. The behavior of the bone model was studied under the same physiological conditions as stated in the first case. The maximum principal stress distribution was obtained. The orthotropic assumption brings the analysis a step closure, if not completely, to the actual anisotropic behavior of the bone.

The maximum principal stress value of cortical section was 0.384 GPa in the neck region. There is a significant decrease in the maximum stress value for the cortical section, because the orthotropic properties show huge variation in directions 1 and 2 (perpendicular to the longitudinal axis). But, still the cortical is stiffer than the cancellous.

The maximum principal stress value of the cancellous section was 0.087GPa situated in the neck region only. Here, there is an increment in the maximum stress value compared to the first case. This is because, the properties are not much varying from isotropic to orthotropic assumption in other two directions (perpendicular to principal direction). And the orthotropic behavior of the cortical bone transfer more load to the cancellous section there by generating more stress in the cancellous neck region.

If we observe the femur bone of a human, the cancellous section is widely distributed and cortical section is thinner in the neck region, and being the weak section of the bone it is vulnerable to fractures. In case of osteoporotic conditions the situation becomes worse and even a small amount of sudden load can break the femur neck.

Chapter 7

Future Study

7.1 Osteoporosis

Osteoporosis is the condition in which the bone becomes weak and brittle. With the age, the regeneration rates of bone gets slower. This leads the cancellous bone to become highly porous, which makes it further weaker. According to a study osteoporosis currently affects over 53 million people in the United States (U.S.) [13].

The most common fracture due to osteoporotic condition is the femoral neck fracture. This is due to the high amount of porous cancellous bone near the neck region of the femur bone. In such cases the patient has to undergo total hip replacement procedure which involves replacing the neck and ball portion of the femur bone with a compatible high strength biomaterial (like Titanium in combination with ceramics).

By defining a new material model considering the porosity of the cancellous section, this case can be well studied to help the physicians understand and predict the possible dangers. The porosity effect the density of the bone, so by providing a relation between elastic properties and density a new material model can be defined.



Fig 7.1 Osteoporosis

7.2 Implant material models

Total hip replacement cases across the globe have increased a lot in number for the past few years. The principal material being used for implants is Titanium which is superior in mechanical properties compared to femur bone. With the implant models the stress distribution will be suppressed to the femur bone which affects the regeneration capacity of the femur bone. The common problem occurring due to above discussed case is, the revision surgery after 10-15 years of the first hip replacement surgery.

By designing a proper implant model and inserting it into the femur bone model can help us understand the dynamics in detail. Further by defining the implant with different material models along with different properties and studying the model can help in designing an artificial material to prevent the difficulties post surgery.

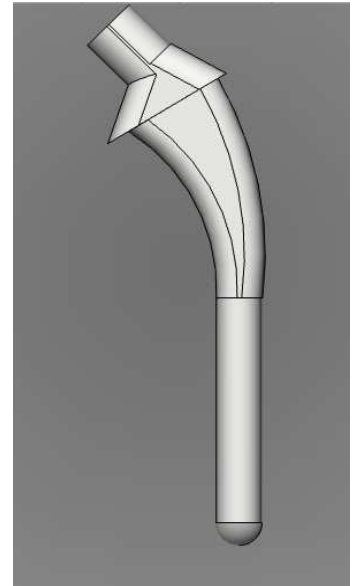


Fig 7.2 Implant model

References:

- [1]. Tsouknidas, A., Savvakis, S., Asaniotis, Y., Anagnostidis, K., Lontos, A., Michailidis, N. (2013) The effect of kyphoplasty parameters on the dynamic load transfer within the lumbar spine considering the response of a bio-realistic spine segment. *Clinical Biomechanics* 28 (9–10), pp. 949–955
- [2]. Finite element analysis of human femur bone. Uzair N. Mughal¹, Hassan A. Khawaja² and M. Moatamedi³. ¹Industrial Engineering Department, Narvik University College (NUC), Narvik, Norway. ²Department of Engineering & Safety, University of Tromsø (UiT), Tromsø, Norway. ³Narvik University College (NUC), Narvik, Norway
- [3]. Zienkiewicz, O.C., R.L. Taylor, and J.Z. Zhu. *The Finite Element Method: Its Basis and Fundamentals* 2005: Elsevier Science.
- [4]. "Finite Element Analysis: How to create a great model". *Coventive Composites*. 2019-03-18. Retrieved 2019-04-05.
- [5]. Modelling of bone fracture and strength at different length scales: a review. Fereshteh A. Sabet, Ahmad Raeisi Najafi, Elham Hamed and Iwona Jasiuk. Department of Mechanical Science and Engineering, University of Illinois at Urbana-Champaign, Urbana, IL 61801, USA
- [6]. Biomechanical Characteristics of the Bone. Antonia Dalla Pria Bankoff, University Of Campinas, Brazil
- [7]. New micro scale constitutive model of human trabecular bone based on depth sensing indentation technique. Marek Pawlikowski a, Krzysztof Jankowski a, Konstanty Skalski b.
- [8]. Behaviour of Human Femur Bone under Bending and Impact Loads. K. V. Arun ¹, K. K. Jadhav ². ¹Department of Mechanical Engineering, Government Engineering College, Haveri, Karnataka, India. ²Union Public Service Commission, New Delhi, India
- [9]. Constitutive modeling of cortical bone considering anisotropic inelasticity and damage evolution. Noritoshi ATSUMI*, Eiichi TANAKA**, Masami IWAMOTO* and Satoko HIRABAYASHI*
- [10]. Biomechanical Analysis of the human femur bone during normal walking and standing up. A.E. Yousif ¹, M.Y. Aziz ².

- [11]. Mechanical Characterization and FE Modelling of a Hyperelastic Material. Majid Shahzad a*, Ali Kamran b, Muhammad Zeeshan Siddiqui a, Muhammad Farhan a
- [12]. Constructing anisotropic finite element model of bone from computed tomography (CT). Siamak Kazembakhshi and Yunhua Luo*
- [13]. <https://www.medicalnewstoday.com/articles/155646.php>
- [14]. <https://melioguide.com/osteoporosis-prevention/neck-of-femur-femoral-neck/>
- [15]. New microscale constitutive model of human trabecular bone based on depth sensing indentation technique. Marek Pawlikowski a, Krzysztof Jankowski a, Konstanty Skalski b
- [16]. Finite Element Analysis of Knee Articular Cartilage. Randall Heydon
- [17]. NON-LINEAR 3D FINITE ELEMENT ANALYSIS OF THE FEMUR BONE Sivakumar V1, Asish Ramesh U. N2
- [18]. Fracture Prediction for the Proximal Femur Using Finite Element Models: Part II—Nonlinear Analysis. J.C. Lotz¹, E. J. Cheal, W. C. Hayes
- [19]. Fracture Prediction for the Proximal Femur Using Finite Element Models: Part I—Linear Analysis. J. C. Lotz¹, E. J. Cheal, W. C. Hayes
- [20]. Unconventional Modeling and Stress Analysis of Femur Bone under Different Boundary Condition. Muhammad Shahzad Masood, Atique Ahmad, Rizwan Alim Mufti
- [21]. An anisotropic visco-hyperelastic model for ligaments at finite strains. Formulation and computational aspects. E. Pen˜ a, B. Calvo, M.A. Martı´nez, M. Doblare
- [22]. Human Tissue Hyperelastic Analysis. Jo˜ao de Sousa Crespo a, Instituto Superior Tecnico, Lisboa Portugal
- [23]. Hyperelastic “bone”: A highly versatile, growth factor – free, osteoregenerative, scalable, and surgically friendly biomaterial. Adam E. Jakus,^{1,2} Alexandra L. Rutz,^{2, 3} Sumanas W. Jordan,⁴ Abhishek Kannan,⁵ Sean M. Mitchel^{1,5}, Chawon Yun,⁵ Katie D. Koube,^{1, 2} Sung C. Yoo,^{1, 2} Herbert E. Whiteley,⁶ Claus-Peter Richter,⁷ Robert D. Galiano,⁴ Wellington K. Hsu,^{2,5} Stuart R. Stock,⁸ Erin L. Hsu,^{2,5} Ramille N . Shah ^{1, 2, 3, 9*}

[24]. EXPERIMENTAL ANALYSIS OF STRESSES IN REAL (PRESERVED) INTACT PROXIMAL HUMAN FEMUR (THIGH) BONE UNDER STATIC LOAD. Maheshkumar V.Jadhav¹, V.R.Gambhire², G.S.Kamble³.

[25]. Understanding the structure–property relationship in cortical bone to design a biomimetic composite
Flavia Libonati, Laura Vergani.