Journal of Basic and Applied Engineering Research
p-ISSN: 2350-0077; e-ISSN: 2350-0255; Volume 6, Issue 4; April-June, 2019, pp. 237-242

http://www.krishisanskriti.org/Publication.html

Modelling and Biomechanical Finite Element Analysis of Human Femur Bone

Ardh Kumar Shukla[1] and Dr. Vinay Pratap Singh[2]

[1]PG Student, HBTU Kanpur
[2]Assistant Professor, HBTU Kanpur
E-mail: [1]ardh.shukla26@gmail.com, [2]vinayforus@gmail.com

Abstract—*Femur being the longest bone of the human body undergoes more deformation under varying load conditions. The failure of the femur bone is the common case which results in partial knee replacement or total knee replacement especially in case of woman. Before going for implantation it is necessary to analyse the implant in case of shape, size, material property, case of failure for which FEA analysis is an effective tool. The aim of this study is to do the FEA analysis of femur bone by creating a real proximal model of the human femur bone.[1-2]*

Biomechanical Finite element modelling is widely used to determine the mechanical behaviour of the long bones which have been created from CT (Computer Tomography) images into an FEA model to find out the stress condition, the mode of failure, need of improvement in design. In this paper, we have developed a three dimensional model of the human femur bone With the increasing case of osteoporosis there is a need to develop implant of femur bone with customized analysis with respect to practical loading conditions. In order to achieve this, we have modelled the FEA model of Femur bone through CT scan which was segmented and analysed through MIMICS software and convert it into a 3D CAD model in Solid work. The model was analysed for cortical bone in comparison to different implant materials [3-5].

Forces normally experienced by humans during daily living activities, also in uncertain cases like an accident, twist, etc. causing femur deformation or failure. Thus, it is necessary to analyse the material properties, structure, load resistance and chance of failure of human femur

The analysis of the above study would help the orthopedic surgeon to analyse the biomechanical behavior of the femur bone and to find the best way to perform the bone prosthesis [6-7]

Keywords: *Biomechanics, CAD Modelling, FEA, FEM, Femur Bone, Ti-6Al-4V, Stress.*

1. INTRODUCTION

Femur bone being the longest bone of the human skeleton system is most important for the human anatomy. The human anatomy of the femur bone is very important to understand the biomechanics of the femoral bone in which upper part of the bone is called as 'head with neck' of the femur bone and part below neck region is known as 'stem or body' of the femur, as shown in Fig. 1. Analysis of femur bone is very critical for the failure analyses of the femoral implant. Total Body weight acts on the femur bone and distinct bone reaction acts on the femoral implant which may lead to failure due to which the analysis of the femoral bone becomes more important.

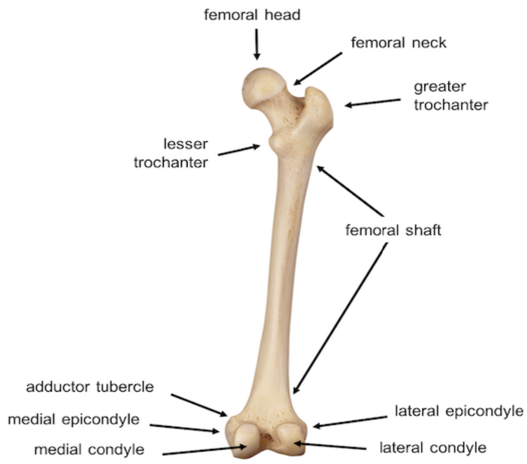

Figure 1 Proximal Femur Bone [3]

Human femur bone bears most of the body weight during daily activities. In this paper, we have modelled the proximal human femur bone in Solid works by the CT data. The solid model was then analysed for various daily activities for cortical bone and Ti-6Al-4V in ANSYS 17.1. This paper involves modelling of complete 3D proximal femur bone from CT scan data. The Solid work software is used to create 3D solid FEA models and segmentation of the smooth surface of the model. The Finite Element Method is used to find the stress and deformation on the femur bone at different load conditions.

2. FINITE ELEMENT METHOD

Finite element method (FEM) is a method widely used for the prediction of stress failure of a dynamic system and as well as biological models. As the demand for medical implants is increasing the prediction of exact failure of such devices is very important to predict the service life of the implant. The different types of finite elements method help to find the effect of loads, their modes of behavior either linear or non-linear, or different types of load application pattern (static, elastic, or dynamic), helps to perform different types of analysis like static structural analysis, modal analysis, fatigue analysis fracture, and material optimization, etc help to carry out the real-time problem simulations with accurate results.

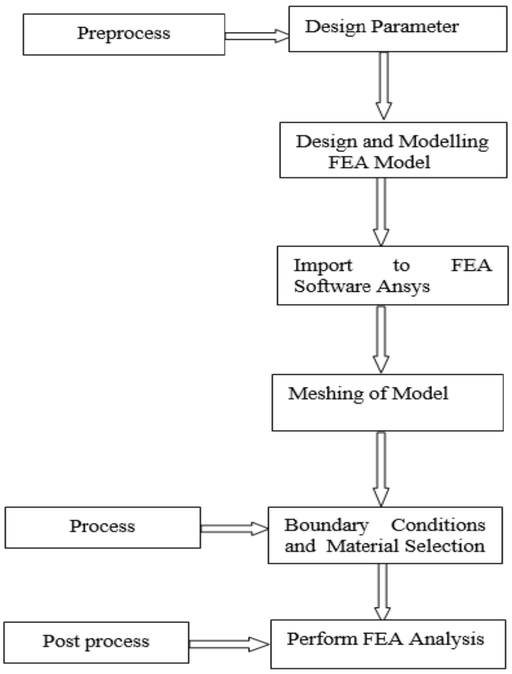

Figure: - 2 FEA Flowchart for analysis of Femur Bone

FEM provides an advantage over the experimental process to find complex data which is necessary to find the optimum working condition for the implant.

- Less implementation cost and involved time are less.
- Measure accurate stress , strain and deformation under varied load conditions
- Used to fabricate virtual design of complex biomechanical design which are difficult to fabricate.

2.1 FEA of Human Femur Bone

FEA is the most commonly used method to find out the static stresses developed in engineering biomechanics. It is widely used to test the life cycle of orthopedic implants and to find the best optimum conditions for their use. It helps to find the stresses in various complex bones which help to design the best implants. It helps to find out the zone of high stress which may lead to the failure of the implant

Human Femur bone is different from human to human in both terms of bone geometry and also in the mechanical properties due to which experimental analysis became very complex.

Table 1:- Dimensions of Femur Bone

Dimensions	Mean Value
Femoral Length	413.4
Femoral Head Offset	44.01
Femoral Head Diameter	43.4
Femoral Head Position	53.1
Neck-Shaft Angle	122.9

So as to analyse the human femur we modelled the artificial model of the femur from CT scan data with identical geometry and same material properties as that of the human femur bone .This FEA model of femur bone help to find out the stress on the human femur bone under varied load condition to obtain the optimal working conditions for the implants .The CAD model of femur modelled in FEA software helps to analyse the stresses and find out the zone of failures of the femoral bone.

The CAD model modelled using Solid work after segmentation from the data obtained from CT scan / MRI is used to build the FEM model. In this paper, we have modelled FEA model of proximal human femur bone.

It is essential to find out the real-time stresses on the femur bone to predict its failure, behaviour under a varied range of bone quality and size. The aim of this paper is to analyse the stresses and deformation acting on the proximal femur bone during daily activities.

3. MATERIALS AND METHODS

The Accurate and precise topology of all the complex bones including femur is an important aspect while creating an accurate 3D model of femur bone which will help the orthopaedic surgeon to planned an effective strategy during femoral implant surgery.

3.1 Methodology

The CAD model of human femur bone is modelled for the study of the forces acting on the proximal femur bone. The complex 3D geometrical model of femur bone is modelled on the basis of CT images into 3D CAD model using Solid work.

3.2 Creating Model of Human Femur Bone

FEA analysis of femur bone is very complex as due to its complex geometrical structure so firstly we create a three dimensional model of femur bone. In this paper we have design a model of femur bone from the CT scan images which are in the form of DICOM image.

Then it is segmented to form 3D model which is then imported into ANSYS 17.1 The model is converted into two parts which include upper part head and neck and lower part which include femur and its stem length . We consider the upper part as cortical bone and lower portion as trabecular bone.

Steps to create 3D model of femur bone:-

- First import the CT images to MIMICS window then it will show the model in geometry window, create a new plane on the head of the femur along the axis of the new plane and rotate it to an angle of 90 degrees to make the direction of cut in X-Y plane.
- The new plane created over the surface is shifted to Z direction towards the distal head of the proximal femur and shifted it to X direction and prepare it to slice the surface to form the 3D model.
- After slicing of femur using freeze command in ANSYS 17.1 by applying slice command on the femur bone to convert the raw surfaces by applying slice command on the femur bone.
- Now femur bone is segmented into two parts including upper part (Head of the femur and the neck) and lower parts of the bone.
- The surface separating the upper part from the lower part is considered as a contact region and calculate stresses in the contact region in the ANSYS workbench 17.1.
- After slicing we get the complete model of femur bone without contact surfaces between the upper and lower part of the bone and applies the properties and assign material to the upper part of the proximal femur bone.

4. CAD MODEL OF FEMUR

Femur CAD model was developed in SolidWorks using the model segmented from CT scan data and converted into FEA model to find the relative stresses in the proximal region of femur bone. The CAD model of the femur bone is segmented and the hollow curves are eliminated to form the solid 3D model.

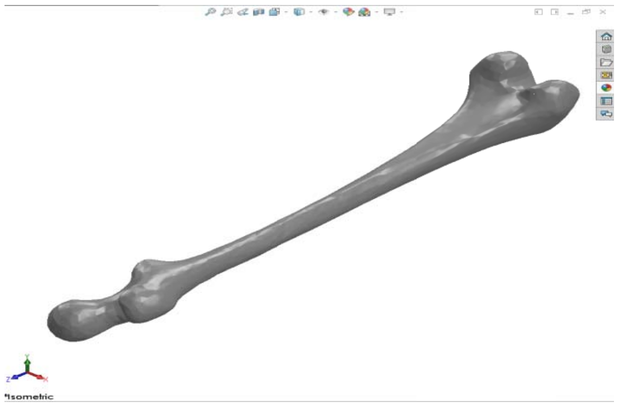

Figure 2 Solid work CAD Model

Creating marrow cavity is very hard to deal about in CAD modelling geometrical description of the cavity is very rare to extract from the CT data.

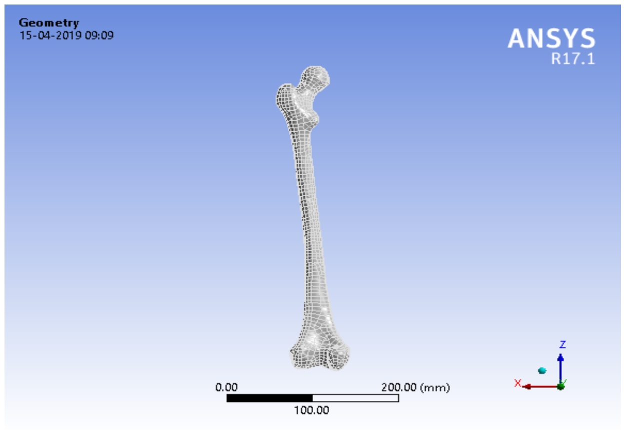

Figure 3 Geometrical CAD Model of Femur Bone

5. CAD MODELLING OF FEMUR BONE

Design Considerations

While modelling the femur bone considering material is very important for simulating the desired zone of stresses**.** As material of the bone is anisotropic not homogeneous so while modelling we consider bone as homogeneous and isotropic within certain limits. As human bone is made up of two kinds of materials compact and spongy resembling so a composite material. We have considered these data while designing a model of femur bone.

FEA analysis of femur bone is done by modelling a three dimensional model of femur was developed in which we have considered the geometrical dimensions of the femur bone. Many studies are performed considering to analyze either a frozen bone or wet bone, synthetic bone or a bone with apparent density but in this study here, we have considered geometrical data of real proximal human femur bone in the form of CT scan DICOM format , to modeled a 3D CAD model. Digital Imaging and Communications in Medicine (DICOM) format contains binary data elements which is segmented to 3D CAD model .CT scan data contains all the geometrical details of the model which is in two-dimensional gray scaled images of a human femur bone. The Hounsfield Unit of the bone corresponding to each element and nodes are converted into gray values and then material properties of bone is analysed with different types of material.

6. ANALYSIS OF HUMAN FEMUR BONE

Meshing

After creating model, for further Finite element analysis (FEA), a tetrahedron surface mesh is generated for femur bone model in ANSYS workbench 17.1. This surface mesh is used to analyse various mesh size with respect to the loading conditions. The volumetric mesh can be generated in ANSYS for the model of femur bone. The FEA software ANSYS 17.1

was used for generating tetrahedron surface mesh to find out the highly stressed zone.

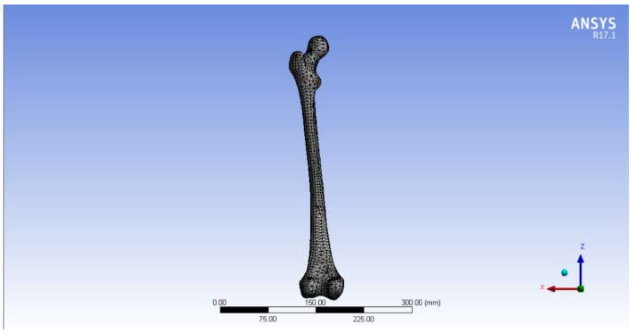

Figure 3: Lateral Mesh Generation on Femur

Mesh Preparation

Meshing is an important part of performing FEA analysis of the CAD model, an optimized mesh is developed using mesh interface in ANSYS Workbench, an optimized set of values is given in the mesh to use smaller elements on proximities and curvatures for the proximal model of femur bone. Mesh refinement is performed to remove the unrealistic stress concentration points to improve accuracy of the results Mesh sensitivity analysis help to ensure the quality of results

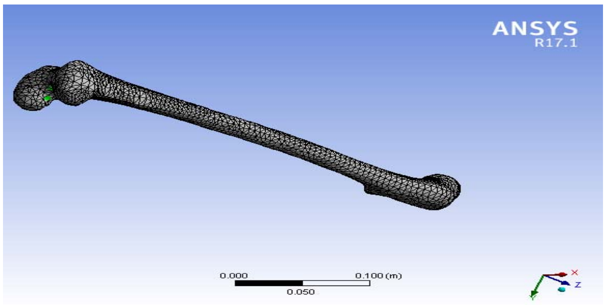

Figure: - 4 Mesh Diagram of femur Bone

7. MATERIAL PROPERTIES

Being heterogeneous and nonlinear in nature, it is very difficult to assign material properties along each direction of the bone model due to which its analysis become complex. In Biomechanical study of the model we can assign material in two ways either in Mimics or during CAD modelling in solid work. We can solve a small portion of bone for anisotropic solution which become extremely difficult to solve for complete bone.

The Young's modulus of the femur bone varies from 10 to 20 GPa and Poisson's ratio and density used were 0.3 and 2000 Kg/m3 respectively Here material properties are directly assigned in ANSYS

Table 2:- Material Properties for FEA Model [4]

Property	Cortical Bone	Ti-6AL-4V
Density(kg/m^3)	1750	4520
Elastic Modulus, E (Gpa)	16.7	113.80
Poisson's ratio	0.3	0.342
Elasticity	Linearly Elastic	Linearly Elastic
Isotropy	Isotropic	Isotropic

7.1 FEA Model Description

The 3D FEA model of femur bone consists of 15588 total numbers of nodes and 8603 numbers of elements in tetrahedral mesh. Three noded linear triangular element is used to element is used to create tetrahedral mesh in ANSYS17.1 having minimum edge length of 3.15640 mm.

Table 3: - Length of proximal femur bone

Direction of Femoral Head	Length (in mm)
XY	86.312
YZ	89.156
ZX	412.11

7.2 Applying Boundary Conditions

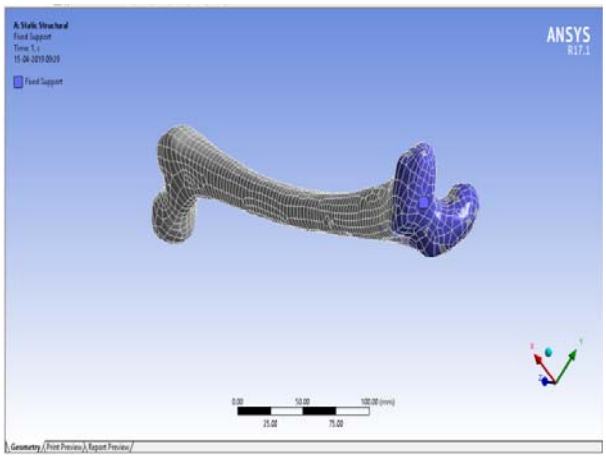

Figure 5:- Application of constraint

1.3 Selection of Area for applying forces

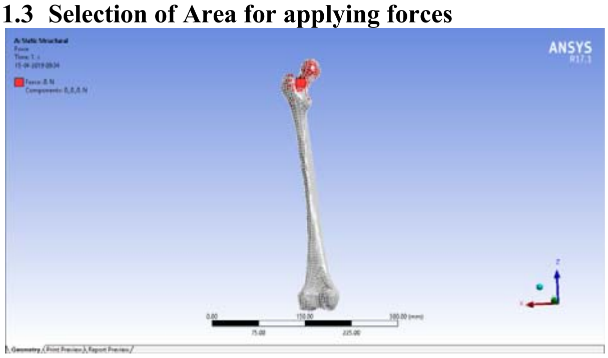

Figure 6:- Applying force constraint

8. RESULT AND DISCUSSSION

This paper analyses the load on the femur bone while performing different daily activities like standing, walking, jumping, and running. In this paper we have analysed the stresses and deformation on the femur bone for cortical bone and Ti-6AL-4V .This analysis will help surgeon to make strategies for preoperative femur surgeries and femoral bone prosthesis.

Table 4:- Load during daily activites [9]

Daily Activites	Load on the Femoral Joint (in N)
Standing	705
Walking	750
Jumping	850
Running	1410

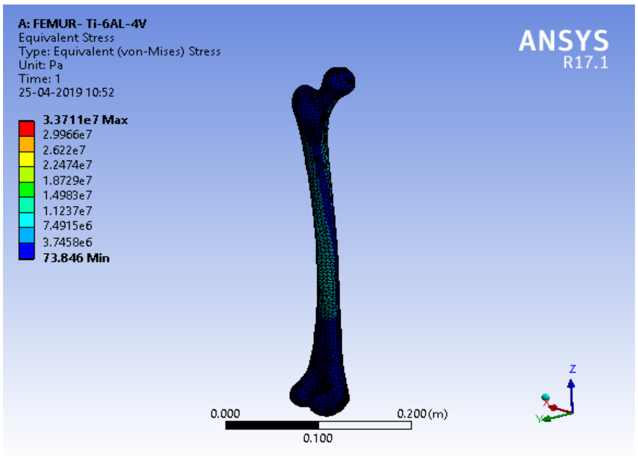

Figure 6:- Von Misses Stress: – Ti-6AL-4V

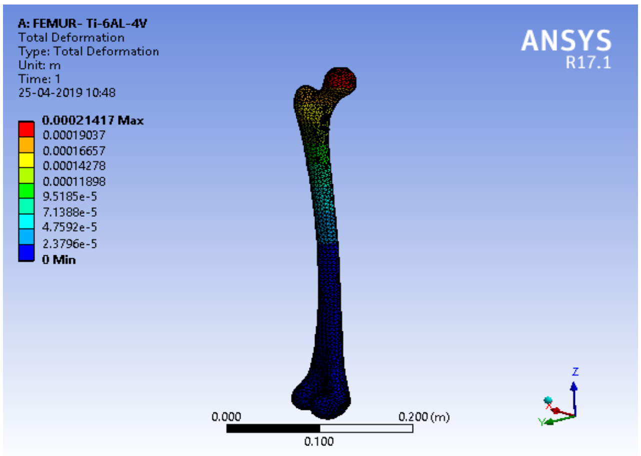

Figure 7 :- Total Deformation: – Ti-6AL-4V

Table 5:-Tabular data representing stresses on Femur bone during daily activities

LOAD	Von-Misses Stress (MPa)	Deformation(m)
705	33.71	0.000214
750	35.65	0.001278
850	40.64	0.002482
1410	63.55	0.003632

Table 6:- Tabular data representing stresses and deformation for Ti-6AL-4V during daily activities.

LOAD (N)	Von-Misses Stress (MPa)	Deformation (m)
705	33.45	0.001265
750	35.60	0.001285
850	41.25	0.002583
1410	53.65	0.003654

9. CONCLUSIONS

In the present work we have modeled 3D CAD model of human femur bone and we have analysed it under different load conditions to predict its failure and high zone of stresses. We conclude that the proximal below neck portion of the human femur bone is subjected to with respect to the head of the femoral stem. We have made comparison between stresses acting on human femur bone cortical bone with respect to the Ti-6Al-4V at different load conditions especially under different daily activities like standing, running, jumping, walking etc.

The analysis under different load conditions gives following result:-

- Comparison is done among Ti-6Al-4V for different daily activities by taking human femur bone analysis as reference for the same load. Table 5 and 6 shows that the result of stresses and deformation is almost same for Ti-6Al-4V. We can say that value of stresses for Ti-6Al-4V is almost same or less as that of the cortical human femur bone. Hence Ti-6Al-4V can be best suitable for femoral implant and orthopaedic implant surgeries.

- Ti alloy posses excellent Bio-compatible properties along with physical properties which makes it an ideal implant material for fractures when compared with natural femur bone

- Ti-6Al-4V is extremely light with less density due to which it become ease for patients in daily activities like walking, standing, running etc.

REFERENCES

[1] Coelho, P. G.; Fernandes, P. R.; Rodrigues, H. C.; Cardoso, J.B. ; Guedes, J.M. (2009) Numerical modelling of bone tissue adaptation—A hierarchical approach for bone apparent density and trabecular structure ,Journal of Biomechanics, 42 830–837

[2] Simulation of the mechanical behaviour of a HIP implant fixed to bone by cementation under arbitrary load-C R Oldani1 and A A Dominguez, Journal of Physics:conference series 90,Pg:142-146.

[3] Comparison of loading behaviour of femural stem of Ti-4Al-6V and cobalt-chromium alloys: a three dimensional finite element analysis-R.R. Clarke, I C Gruen, Sarmiento and pathologies.lexmedicus.com.au/pathologies/hip-avascular-necrosis

[4] T.A. Brown, L.Kohan, B.Ben-Nissan ―Assessment By Finite Element Analyis of the Impact of Osteoporosis and Osteoarthritis on Hip Resurfacing‖, 5th ACAM 12-17 dec 2007.

[5] TOMMASINI S. M., NASSER P., SCHAFFLER M. B., JEPSEN K. J.Relationship between bone morphology and bone quality in male tibias: implications for stress fracture risk, J Bone Miner Res, 2005, 20:1372–1380.

[6] BENNELL K., MATHESON G., MEEUWISSE W., BRUKNER P., Risk factors for stress fractures, Sports Med, 1999, 28:91–122.

[7] GILADI M., MILGROM C., SIMKIN A., DANON Y., Stress fractures. Identifiable risk factors, Am J Sports Med, 1991, 19:647–652

[8] Aligner, Weipert A. Designing principles of custom-made hip stems. Proc 3rd annual symposium on custom-made prostheses, Nice, 1990

[9]Nithin Kumar KC, Tushar Tandon, Praveen Silori, Amir Shaikh Biomechanical stress analysis of a Human Femur bone using ANSYS 4th International Conference on Materials Processing and Characterization Materials Today: Proceedings 2 (2015) 2115 – 2120.

[10]Simulation of the mechanical behaviour of a HIP implant fixed to bone by cementation under arbitrary load-C R Oldani1 and A A Dom inguez, Journal of Physics: conference series 90, Pg: 142-146.

[11] Comparison of loading behaviour of femural stem of Ti-4Al-6V and cobalt-chromium alloys: a three dimensional finite element analysis-R.R. Clarke, I C Gruen, Sarmiento.

[12] T.A. Brown, L.Kohan, B.Ben-Nissan ―Assessment By Finite Element Analyis Of The Impact Of Osteoporosis And Osteoarthritis on Hip Resurfacing‖ , 5th ACAM 12 -17 dec2007.

[13] Zienkiewicz, O.C., R.L. Taylor, and J.Z. Zhu, The Finite Element Method: Its Basis and Fundamentals: Its Basis and Fundamentals 2005: Elsevier Science.

[14] Stolarski, T., Y. Nakasone, and S. Yoshimoto, Engineering Analysis with ANSYS Software 2011: Elsevier Science.