Abstract: In recent years, studying behavior of human bone against various loadings with finite element simulations has been cost effective alternative to experimental methods. This study requires a computer model which can be obtained either from CT scan data or MRI images. The present paper deals with creating CAD model of a human femur of a human body and in turn stresses and deformation are obtained for various static loading conditions. The result is validated with published research paper and convergence test is conducted. The results obtained are conforming within the engineering errors limit. The algorithm can provide a tool to extract quantitative information of human anatomical structure. Therefore, the demonstrated results can be applicable to solve a specific biomechanical problem of human.

Keywords: human femur, stress analysis, CAD modeling

I. INTRODUCTION

Biomedical engineering includes knowledge for the development and advancement of new technology at the interface of engineering, biology and medicine.

Working in biomedical engineering, applies engineering approaches to understand living systems, develop methods to repair and replace damaged or diseased organs, measure the internal structures of human body in health and disease, develop new diagnostic tools, and create many ways to make life healthier and safer.

Efforts are made to reduce the disability in human organs by artificially implanting the organs. Computer Aided Design (CAD) is trying to develop the models, which are capable of simulation, analysis and manufacture of human organ implants.

Accurate 3-dimensional models of bone are required in several biomedical applications. Three dimensional finite element analyses has been widely used to study the biomechanical behavior of the bone in academic research and clinical applications such as bone remodeling, interaction of bone and muscle, design of implants and evaluation of fracture risk. In order to simulate the reactions of bone to physiological and non-physiological loading conditions multiple computer aided finite element model of femur can be developed. The development of CAD model can be done by using Computed Tomography (CT) and Magnetic Resonance Imaging. In general CT/MRI images yield detailed three dimensional body structures through a series of cross-sectional images of human body organs which is used for diagnostic and 3D visualization purposes in medical profession. CAD coupled with medical imaging and Rapid prototyping technologies can help in making realistic models of human body organs for clinical purposes.

The integration of medical imaging, CAD and numerical tools like Finite Element Methods (FEM) has provided a new advancement in the field of human body modeling and analysis. The Computed tomography (CT) is currently the gold standard for the acquisition of data from which the 3D models of human organs are generated. CT data can provide the accurate geometrical topology and material properties of bone.

II. HUMAN BODY MODELING TECHNIQUE

In general the human body models are broadly classified in two categories as physical models and numerical models.

Physical models

These are rigid crash dummy models and are widely used in evaluation of protective measures. These are made of tissue mimicking materials and resemble the real human anatomy.
Numerical models

These are prepared by using imaging technique like CT/MRI and laser scanners. The data so captured is rebuilt to create a model which is called a numerical model. This is also known as Reverse Engineering (RE) process to develop a model. Finite Element Method is a numerical method development technique.

The Finite element technique for modeling of the human organs can be cost effective alternative as compared to physical model. So the numerical models can be created to simulate the effect of loads on human body organs. CT/MRI data can be used to get a realistic and reliable FE heterogeneous human body model for biomechanical analysis and simulation along with automatic tetrahedral mesh generation.

In this work CT scan data are used to input to create the FE model for femur bone. The material properties are provided as the function of Hounsfield Unit (HU) which is calculated from the CT scan data. Hounsfield Unit represents the numeric information contained in each pixel of a CT image. It is related to the composition and nature of the tissue imaged and is used to represent the density of tissue. It is also called as CT number. Each voxel is assigned a value on a scale in which air has a value of −1000; water, and compact bone, +1000. The logical flow of present approach is shown in Fig 1

![Flow diagram of image based human organ FE modeling and analysis](image)

The present study is carried by developing of FE model of human femur using CT scan data and providing heterogeneous material properties based on Hounsfield Unit. Then the stress and deformation of the bone is calculated based on various loading conditions.

The present work is aimed in creating a FE model of human femur bone from CT scan data. The representation schemes used for material modeling, reverse engineering and its application for human body modeling.

Kohet al.[16] provided X-ray-based method for reconstruction of femur bone with two X-ray images and three CT images. The obtained femur model is closer to a CT-based 3D femur model in comparison with the reconstruction method using only X-ray images.

Yoshibash et al. [33] presented a high-order Finite element method (FEM) for simulating the bone response to loads. An accurate model of the bone geometry was constructed from a quantitative computerized tomography (QCT) scan using smooth surfaces for both the cortical and trabecular regions. Inhomogeneous isotropic elastic properties are assigned to the Finite element model using distinct continuous spatial fields for each region. The Young modulus is represented as a continuous density function computed by a least mean squares method.

Then Filippi et al. [10] studied and the development of a script for a commercial software package (3ds Max) to reconfigure the model of a femur from two orthogonal images.

Gamage et al.[11] compared the model of femur generated from the X-ray data and CT scan data. The modeling was done using a bi-planar shape customization framework combined with several image processing methodologies can yield acceptable results for patient-specific femur reconstruction.

Kirana and Ghosal [15] presented a procedure for the reconstruction of biological organ from image sequence obtained through the CT-scan. The procedure presented used only free software available instead of working on commercial software.

Pise et al. [25] presented a B-spline based modeling and mesh generation using CT scan images for bio-object. The distinct advantage of this method is that the modeling and the meshing are done in a single step along with true representation of model.

Raji and Veerendra [22] developed a finite element model of femur using CT scan data with material properties retrieved from the CT scan data.

In the present study, the modeling of human body femur is done by using the CT scan images from the input CT scan data the solid model of femur is developed which is further meshed for FE analysis. The model developed is meshed with tetrahedral elements. For providing the material properties the CT data is calculated and the properties are provided as the...
function of CT data. The study is very useful for orthopedic organ transplantation of human organs and analyses under physiological loading conditions.

III. MATERIALS AND METHODS

First, for the FE analysis of the femur, the first step is the development of the three-dimensional model of the bone. In this study the model is generated using 2-D CT scan slices. The 2-D slices are converted into 3-D model by processing the data into medical based image processing software MIMICS 9.11. Materialise's Interactive Medical Image Control System (MIMICS) interfaces between scanner data (CT, MRI) and Rapid Prototyping, CAD and Finite Element analysis. It is an image-processing package with 3D visualization functions that interfaces with all scanner formats.

The detailed explanation for the model generation is given below:

A. CT data

To generate the 3-D/CAD model of a biological object, CT (Computed Tomography) data or the MRI (Magnetic Resonance Imaging) data are the primary source of input data. In the present study, the CAD model is generated using CT data of the biological object i.e. Femur in this thesis. The data is obtained in slice by slice and processed into MIMICS (Materialize Interactive Medical Imaging Control System). The CT slices are in DICOM (Digital Imaging and Communications in Medicine) format which is the standard file format for handling, storing, printing and transmitting information in medical imaging [32].

For the present study, the data used in analysis belongs to a 30 year old male with following parameters: 140 kV, 250 mA and 0.75 mm slice thickness, axial scan without overlap with pixel size of 0.78 mm. The total CT scan data contains 97 slices out of which 3 have been neglected from distal end.

B. Model development methodology

Present effort describes about the modeling of the human femur bone from CT scan data using reverse engineering technique. The mesh is prepared from the CT scan slice having tetrahedral elements. The CT scan slices are processed in MIMICS 9.11 for generation of meshed 3D finite element model.
C. MATERIAL ASSIGNMENT

The modulus of elasticity and density i.e. E-ρ relation for the human femur is based on the HU value. The assumption for the FE model in the present study is that the cortical and trabecular bone are not distinguished. The same material properties are assigned on both the regions.

DENSITY ASSIGNMENT

Based on the Hounsfield Unit (HU), the bone material properties could be obtained through apparent density. The apparent density (ρ) is defined as the density without fluid influence. For dense cortical bone, the relationship between the apparent density (ρ) and the Hounsfield number (HU) is given by \[ \rho = 1 + 7.185 \times 10^{-4} \times HU \]

YOUNG’S MODULUS ASSIGNMENT

The young’s modulus of the bone is a function of apparent density. Therefore, bone is considered as an inhomogeneous material. The relationship between the apparent density and the young’s modulus varies with the direction of load as axial and transverse loading condition [29]. For the case of axial loading the functional relationship is approximated by \[ E = 2065 \rho^{0.29} \] and in case of transverse loading the relationship is \[ E = 2314 \rho^{1.57} \] where E is Young’s modulus (MPa) and ρ is apparent density (g/cm³).

In this study the axial loading is considered on the human femur bone.

POISSON’S RATIO ASSIGNMENT

Concerning Poisson’s ratio of cortical and cancellous bone different data is available in the literature. The cited value ranges from 0.2 to 0.5 with an average of 0.3 for cortical bone. So the average value of the Poisson’s ratio is used for providing material properties. No details have been given about any correlation with the bone density.

IV. CONVERGENCE OF FE FEMUR MODEL

To understand the behavior of finite element model of femur, a convergence test is performed on femur model. 9 FE models have been generated by varying mesh density in the range of 12379 to 106240 nodes. The FE analysis of all the 9 models are carried with double legged stance loading of 3000 N (vertical load) and boundary condition as explained in Fig 4.

Two boundary conditions are imposed on the bone model. The base is fixed ended and the head is imposed with external point load. On the basis of load application the deformation for all the 9 models is calculated and convergence test is performed

Fig. 4 Representation of boundary conditions for vertical loading

Based on analysis result, one dimensionless parameter is adopted for the convergence test, \[ \Delta U_0 = \left| \frac{U_i - U_{ref}}{U_{ref}} \right| \]

Parameter \( \Delta U_0 \) represent change in deformation of \( U_i \) with respect to \( U_{ref} \) deformation, \( U_i \) is the analysis result of the \( i^{th} \) number of nodes. The relationship between number of nodes and \( \Delta U_0 \) is shown in fig 5. A convergence curve is obtained for \( \Delta U_0 \). The magnitude of \( \Delta U_0 \) is changing continuously while the mesh is refined and number of nodes increases from 12379 to 106240. From the figure it can be concluded that the FE mesh of femur model gets optimized between 78855 to 106240 nodes.

![Fig.5 Relationship between values of \( \Delta U_0 \) and number of nodes](image)

V. LOADING CONDITION:

After the generation of the finite element models with the assignment of bone material properties to finite element mesh different loading conditions were considered.

A. VARYING ANGLE OF LOADING:
On the femur bone the concentric load is applied at the femoral head. The angular direction of load is varied from \(0^\circ\) to \(15^\circ\). On the femoral head of the femur the concentric load of 3000 N is applied and the distal end is constrained as fixed end. Fig 6 shows the application of load and fixed end on femur. With the varying angle of application of force the deformation of the bone is studied.

![Fig. 6 Loading condition for varying inclination](image)

**B. SINGLE LEG AND DOUBLE LEG STANDING**

Many studies show that femur bone experiences a set of forces through gluteal muscle, iliotibial tract and reaction force on femur head. According to physiological condition, for single leg the femur is capable of holding around 3 times the bodyweight. For the case of single leg, the force on the femoral head with joint reaction was estimated as 3.0 times of body weight and the force on greater trochanter through gluteal abductor was 2.2 times body weight. The distal end of femur is constrained.

For the case of double leg standing, the force is applied only on the femoral head as 3.0 times of bodyweight.

For the present study the ct data is of 30 year male human. Therefore, to simulate the loading condition for single and double leg standing in present study, it is assumed that average body weight of 25 percentile of other race person of 30 year male as 638 N[30].

The loading condition for the single leg and double leg conditions are shown in Fig 7 and Fig 8 respectively.

![Fig. 7 loading and boundary condition for femur in single legged](image)

<table>
<thead>
<tr>
<th>(R_x)</th>
<th>(R_y)</th>
<th>(R_z)</th>
<th>(I_x)</th>
<th>(I_y)</th>
<th>(I_z)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-343</td>
<td>331</td>
<td>-1847</td>
<td>343</td>
<td>-331</td>
<td>1315</td>
</tr>
</tbody>
</table>

![Fig. 8 Loading and boundary condition for double leg](image)

<table>
<thead>
<tr>
<th>(R_x)</th>
<th>(R_y)</th>
<th>(R_z)</th>
</tr>
</thead>
<tbody>
<tr>
<td>-343</td>
<td>331</td>
<td>-1847</td>
</tr>
</tbody>
</table>

**C. WALKING WITH VARYING BODY WEIGHT**

During the horizontal walking by humans there are number of forces and reaction that acts on the leg. For the femur bone the
forces and reaction that acts on the bone are such as: joint reaction force, abductors and iliotibial tract.

The joint reaction force is acting due to the hip joint on the femoral head. Due to the hip joint with femur, a force acts on the femoral head which is shown in figure above. The abductor reaction occurs due to thigh muscle. The abductor reaction acts on the greater trochanter of the femur bone. Due to the movement of the bone the reaction force acts on the knee joint. The knee reaction acts as iliotibial-tract on the bone.

For a terminal stance during horizontal walking for weight of 70, 100 and 200 kg the load acting on the femur bone are represented by (I), (II), and (III) respectively[2].

- **(I)** Joint reaction force:
  - \( F_1 = 234 \) N, \( F_2 = 385 \) N, \( F_3 = -1652 \) N
  - Abductors:
    - \( F_1 = 0 \) N, \( F_2 = 0.8 \) N, \( F_3 = -1.937 \) N
  - Iliotibial tract:
    - \( F_1 = 0 \) N, \( F_2 = 0 \) N, \( F_3 = 350 \) N

- **(II)** Joint reaction force:
  - \( F_1 = -334.8 \) N, \( F_2 = -550 \) N, \( F_3 = -2360 \) N
  - Abductors:
    - \( F_1 = 0 \) N, \( F_2 = 1.142 \) N, \( F_3 = -2.8 \) N
  - Iliotibial tract:
    - \( F_1 = 0 \) N, \( F_2 = 0 \) N, \( F_3 = 500 \) N

- **(III)** Joint reaction force:
  - \( F_1 = 669.6 \) N, \( F_2 = 1100 \) N, \( F_3 = 4729 \) N
  - Abductors:
    - \( F_1 = 0 \) N, \( F_2 = 2.285 \) N, \( F_3 = -5.6 \) N
  - Iliotibial tract:
    - \( F_1 = 0 \) N, \( F_2 = 0 \) N, \( F_3 = 1000 \) N

In order to simulate the stress field for the femur the boundary conditions were provided as, on the distal head the displacement is constrained and the external load is imposed.

After the generation of CAD model of human femur, the boundary conditions were imposed on the model to simulate the forces acting on the model for various conditions.

**VI. RESULTS**

**A. VARYING ANGLE OF LOAD**

The relationship between the deformation and the angle of load is plotted and it has been found that with increment in angle the deformation decreases linearly. The result is shown in fig 11.

When the angle of load was varied from 0° to 15°, the effect of variation of deformation is studied with varying angle. From the obtained deformed value for the variation in angle it has been found that with increasing the angle, the deformation decreases i.e. when the load was acting at 0° inclination with respect to Z-axis the deformation is maximum and when the same load is applied at an angle of 15° the deformation decreased. The deformation decrement was linear with linear increment of angle with Z-axis.

![Fig.11 Relationship between deformation and the angle of inclination](image-url)
B. SINGLE LEG AND DOUBLE LEG STANDING

When the loading was applied for single leg standing on the femur bone the maximum deformation of the bone occurred at the femoral head with a magnitude of 0.080619 mm and the lowest is obtained at the lower end. Fig 12 shows the deformation of the femur bone with single leg standing.

For the case of double leg standing also the maximum deformation was found at the femoral head with the magnitude of 0.054134 mm and the lowest at the lower end of the bone. Fig.13 represents the deformation of the bone for the double leg standing.

Fig.14 shows the deformation for the single leg and the double leg. From the plotted graph it has been found that at the distal end (fixed end) the deformation is zero for both cases and increases till the femoral head. The deformation occurred in single leg is more as compared to the double leg for the same body force. Since in case of the single leg standing the forces act on the femoral head and greater trochanter whereas in case of double leg standing the force act only on the femoral head. Due to the absence of load on the greater trochanter less deformation is calculated for the double leg standing.

The von-mises stress maximum value for the case of single leg standing is found to be 1.8013e8 Pa and the minimum value is found as 0.20475 Pa, whereas for the case of double leg standing the von-mises stress maximum value gets decreased and becomes 1.7522e8 Pa and the minimum value reaches 0.21023 Pa. For the case of single leg standing forces are applied to the femoral head and greater trochanter in femur which tend to create more von-mises stress distribution on the shaft of femur whereas in case of double leg standing the greater trochanter force is absent, the load is applied only on the femoral head which causes less von-mises stress generation on the shaft of the femur bone as compared to single leg standing.

The simulated results for the principal stress calculation as: For the case of single leg standing the maximum and minimum stress value are 6.8709e7 Pa and -3.5471e7 Pa respectively, and when the loading conditions are confined to double leg standing the maximum and minimum principal stress value are found as 6.8711e7 Pa and -3.5471e7 Pa respectively. Though the maximum and minimum value for the maximum principal stress is same but the stress...
distribution is different in both cases which can be seen in figure 16. In case of single leg standing more principal stress is generated on the shaft of the femur bone as compared to the double leg standing loading. The von-mises stress and principal stress distribution for single leg and double leg standing are shown in fig 15 and fig.16 respectively.

![Fig.15 Von-mises stress distribution](image)

**Fig.15** Von-mises stress distribution (a) for single leg; (b) for double leg

![Fig.16 Maximum principal stress distribution](image)

**Fig.16** Maximum principal stress distribution for (a) single leg; and (b) double leg

C WALKING WITH VARYING BODYWEIGHT

When the weight of the human body is increased during the horizontal walking condition, at an instance the deformation for the weight of 70 kg, 100 kg and 200 kg are simulated. The result concluded that the deformation of the bone is more for more weight. The maximum deformation for the entire load was found at the femoral head and the lowest at the distal end.

![Fig.17 Deformation](image)

**Fig.17** Deformation for 70 kg, 100 kg and 200 kg in horizontal walking

The deformation for 70 kg, 100 kg and 200 kg are shown in Fig 5.8, Fig 5.9 and Fig 5.10 respectively.

![Fig.18 Deformation](image)

**Fig.18** Deformation for 70 kg

![Fig.19 Deformation](image)

**Fig.19** Deformation for 100 kg
When the weight of the body is increased the simulated results concluded that with increase in weight the maximum deformation, maximum principal stress and von-mises stress increases.

For the weight of 70 kg, 100 kg and 200 kg the evaluated values for von-mises stress, maximum principal stress and maximum principal strain were found as

<table>
<thead>
<tr>
<th>Body Weight (kg)</th>
<th>Von-mises Stress (max) (Pa)</th>
<th>Von-mises Stress (min) (Pa)</th>
<th>Maximum Principal Stress (max) (Pa)</th>
<th>Maximum Principal Stress (min) (Pa)</th>
<th>Maximum Principal Strain (max) (m/m)</th>
<th>Maximum Principal Strain (min) (m/m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>70</td>
<td>1.5457e8</td>
<td>0.042104</td>
<td>6.0225e7</td>
<td>-2.4517e7</td>
<td>-9.234e-9</td>
<td>-1.3043e-8</td>
</tr>
<tr>
<td>100</td>
<td>2.081e8</td>
<td>0.060253</td>
<td>8.6033e7</td>
<td>-3.5029e7</td>
<td>0.00036081</td>
<td>-1.3093e-8</td>
</tr>
<tr>
<td>200</td>
<td>4.4163e8</td>
<td>0.12057</td>
<td>1.7207e8</td>
<td>-7.0056e7</td>
<td>0.00051545</td>
<td>-2.6086e-8</td>
</tr>
</tbody>
</table>

The von mises stress, maximum principal stress and maximum principal strain distribution on human femoral for 70 kg, 100 kg and 200 kg are shown in Fig 21, Fig 22 and Fig 23 respectively.
VII. CONCLUSIONS

The aim of this study is to generate CT scan based FE model of human femur and analyze the stress and deformation occurring in the human femur under physiological loading conditions.

The forces acting on the femur bone with different loading conditions as: by varying the angle of load, single and double leg standing and horizontal walking for different body weight is studied.

For the varying angle of loading it has been concluded that with the increment in angle of load the deformation in the femur head decreases. In biological human femur the loading angle on femur head is around 7-12°. The relationship of the different angle of loading and deformation is studied which is useful in implanting the artificial bone.

The effect of single leg and double leg standing is considered. The deformation and stresses generation are more in human femur for single leg standing as compared to the double leg. Also the influence of body weight on the human femur is studied during horizontal walking. At a particular stance during horizontal walking the deformation and stresses generated in human femur for 70 kg, 100 kg and 200 kg is studied. The simulated results concluded that deformation and stress increases with increase in body weight force and reactions acting.

These simulated results are useful for clinical applications to orthopedic surgeon in replacement of human femur.

References

[4] Antonio T S; Ciaccia M; Muller-Karger C; Casanova E: Orientation of orthotropic material properties in a femur FE model: A method based on the principal stresses directions, Medical Engineering & Physics (2011). (manuscript accepted)
[9] Fedida R; Yosibash Z; Milgrom C; Joskowicz L: Femur mechanical simulation using High-order FE analysis with continuous Mechanical properties, II International conference on computational bioengineering.

Kosivumaki J E M; Thevenot J; Pulkkinen P; Kuhn V; Link T M; Eckstein F; Jamsa T; CT-based finite element models can be used to estimate experimentally measured failure loads in the proximal femur, Bone 50 (2012) 824–829.

Lengsfeld M; Schmitt J; Alter P; Kaminsky J; Leppek R; Comparison of geometry-based and CT voxel-based finite element modelling and experimental validation; Medical Engineering & Physics 20 (1998) 515–522.

Lengsfeld M; Kaminsky J; Merz B; Franke R P; Sensitivity of femoral strain pattern analyses to resultant and muscle forces at the hip joint, Medical Engineering & Physics. Vol. 18 (1996) 70-78.

Materialise’s Interactive Medical Imaging Control System (MIMICS) 9.11: User manual.

McNamara B P; Cristofolini T L; Toni A; Taylor D; Relationship between bone-prosthesis bonding and load transfer in total hip reconstruction, journal of Biomechanics, Vol. 30. No. 6, (1997) 621-630.


Peng L; Bai J; Zeng X; Zhou Y; Comparison of isotropic and orthotropic material property assignments on femoral finite element models under two loading conditions; Medical Engineering & Physics 28 (2006) 227–233.

Pise U V; Bhatt A D; Srivastava R K; Influence of isotropic and orthotropic relationships on the accuracy of B-spline based heterogeneous graded finite element modeling of proximal femur, Computer-Aided Design & Applications, 8(a), 2011


Tuan H S; Hutmacher D W; Application of micro CT and computation modelling in bone tissue engineering; Computer-Aided Design 37 (2005) 1151–1161.

Viceconti M; Bellingeri L; Cristofolini L; Toni A; A comparative study on different methods of automatic mesh generation of human femurs; Medical Engineering & Physics 20 (1998) 1–10.

Wirtz DC; Schilers N; Pandorf T; Rademaker K; Weichert D; Forst R; Critical evaluation of known bone material properties to realize anisotropic FE-simulation of the proximal femur, Journal of Biomechanics 33 (2000) 1325–1330.


www.healthline.com/human-body-maps/femur

www.medical.nema.org

Yosibash Z; RoyPadan R; Joskowicz L; Milgrom C; A CT-based high-order finite element analysis of the human proximal femur compared to in-vitro experiments.

Yosibash Z; Trabelsia N; Milgrom C; Reliable simulations of the human proximal femur by high-order finite element analysis validated by experimental observations, Journal of Biomechanics 40 (2007) 3688–3699.