BIO-MODELING AND BIOMECHANICS OF ILIUM AND FEMUR BONES
K Michael Jai Kumar, N V S Shankar, K Sai Chandu and V Sai Venkatesh
Department of Mechanical Engineering, Swarnandhra College of Engineering and Technology, Seetharampuram, AP. India.

ABSTRACT
Biomechanics is the study of mechanics of biological systems. This is important so as to understand the mechanical behavior of various body parts so that various types of implants, suits, etc., can be easily designed. 3D printing technology is helping in giving better insight to the interactions of various bones. This can be done quickly through biomodelling and printing the same. This paper highlights the use of 3D printing in bio-modelling. Two bones, Ilium and Femur are 3D Printed. FEA is performed on these two bones so as to understand the response of the bones when subjected to different body weights.

Key words: 3D Printing, Biomechanics, Femur, Ilium.


1. INTRODUCTION
Biomechanics deals with the mechanical behavior of biological systems. This is an important field as it helps in understand the structural function of each organ/part of a biological system and thus paves way for developing prosthetics, splints and other systems. Antonia Dalla Pria Bankoff [1] when detailing the biomechanical characteristics of bone, indicated that the minerals calcium and phosphate, along with collagen constitute to 60% to 70% of the bone weight while the rest is of water. These minerals are responsible for the bone strength. Various types of loading of bone were described by him. Tony M Keaveny, et al [2] detailed the mechanical properties of various bones like density, elastic modulus etc. The anisotropic properties of Femoral Cortical bone as detailed by them are listed in table 1. Mechanical properties of various bones are summarized in table 2. Giuseppe Maida[3] discussed how the mechanical problems like motion preservation, shock absorption during spinal surgery can be dealt with using “molla a tazza” (“spring-cup”). Susan Hueston, et al [4] demonstrated how FEM of cervical spline can help in designing implants.
Table 1: Mechanical Properties of Femoral Bone [2]

<table>
<thead>
<tr>
<th>Property</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Longitudinal Modulus (MPa)</td>
<td>17900</td>
</tr>
<tr>
<td>Transverse Modulus (MPa)</td>
<td>10100</td>
</tr>
<tr>
<td>Shear Modulus (MPa)</td>
<td>3300</td>
</tr>
<tr>
<td>Longitudinal Poisson's Ratio</td>
<td>0.4</td>
</tr>
<tr>
<td>Transverse Poisson's Ratio</td>
<td>0.62</td>
</tr>
<tr>
<td>Longitudinal (MPa)</td>
<td></td>
</tr>
<tr>
<td>Tension</td>
<td>135</td>
</tr>
<tr>
<td>Compression</td>
<td>205</td>
</tr>
<tr>
<td>Transverse (MPa)</td>
<td></td>
</tr>
<tr>
<td>Tension</td>
<td>53</td>
</tr>
<tr>
<td>Compression</td>
<td>131</td>
</tr>
<tr>
<td>Shear (MPa)</td>
<td>65</td>
</tr>
</tbody>
</table>

Table 2: Mechanical Properties of various bones [2]

<table>
<thead>
<tr>
<th>Property</th>
<th>Young's Modulus (MPa)</th>
<th>Poisson's Ratio</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cortical Bone</td>
<td>17000</td>
<td>0.3</td>
</tr>
<tr>
<td>Trabecular Bone</td>
<td>70</td>
<td>0.2</td>
</tr>
<tr>
<td>Subchondral Bone</td>
<td>2000</td>
<td>0.3</td>
</tr>
<tr>
<td>Acetabular Cartilage</td>
<td>15</td>
<td>0.45</td>
</tr>
<tr>
<td>Sacroiliac Joint</td>
<td>15</td>
<td>0.45</td>
</tr>
<tr>
<td>Pubic Symphysis</td>
<td>15</td>
<td>0.45</td>
</tr>
<tr>
<td>Reconstruction Plate</td>
<td>200000</td>
<td>0.32</td>
</tr>
<tr>
<td>Sacral Mount</td>
<td>69000</td>
<td>0.35</td>
</tr>
<tr>
<td>Turnbuckle</td>
<td>200000</td>
<td>0.32</td>
</tr>
<tr>
<td>Load Cable</td>
<td>200000</td>
<td>0.32</td>
</tr>
<tr>
<td>Cement (PMMA)</td>
<td>2200</td>
<td>0.34</td>
</tr>
</tbody>
</table>

Shirish [5] used MIMICS software to extract 3D Model of jaw and performed FEA using ANSYS with various bite forces so as to understand the stress distribution during loading of the bone. Chih-Han Chang, et al[6] used FEM in understanding the performance of chin bar of helmet during impact. Maximum acceleration and Head Injury Criterion (HIC) were employed during this study. The experimentation carried out by Ron Jadischke, et al[7] indicated that lower head acceleration is experienced with helmets than without helmets. Haojie Mao, et al [8] performed 35 FE Analyses to accurately model and predict the types of head injuries. They also summarized the mechanical properties of various parts of the head.

Sofia Brorsson [9] discussed the rehabilitation of hand effected by Rheumatoid Arthritis. Various therapies like use of electricity, injections and exercises were detailed.

Femur is the longest and high load bearing bone in human body. It is the bone possessing the largest strength and the most effected during osteoporosis. Thus, understanding the response of Femur is very much important. Seth Gilchrist, et al [10] developed an experimental technique to study the effect of side fall on femoral bone. Digital Image Correlation was used by them to validate their experimental technique. The schematics of the fall simulator used are explained in detail. Taeyong Lee, et al [11] proposed a non-invasive QCT based model for predicting the fracture load and verified the same experimentally. The loading procedure during experimental testing is detailed in their article. Samuel SANCHEZ-CABALLERO, et al [12] detailed a test rig using which the behaviour of a repaired femur is assessed. During the evaluation, artificial femur testing is done to predict the behaviour of original. David J Merriman, et al [13] experimentally investigated the size effect of IM reaming on Femur. As a part of this testing, the experimental investigations were carried by them into Axial Compressive, torsional loading and four-point bending. Experimentation is performed to analyse fracture load.
Figure 1 Loading of FEMU in various types of tests as detailed in [13]

García, et al [14] used FE analysis in understanding the efficacy of pelvis fixators for stabilizing pelvic ring disruptions. Zuoping Li, et al [15] used numerical simulations and experimental studies to investigate the effect of cement filling on the mechanical behaviour of Pelvis. Costin D Untaroiu, et al [16] used FE simulations for estimating whole body response of a mid-sized male human during car-to-pedestrian accidents. He validated the lower limb-pelvis and lumbar spine regions of the human model against the post-mortem human surrogate test data recorded. Zhixiu Hao, et al [17] developed three FE models to study the response of pelvic bone. For this the bone material anisotropy is computed based on CT grey value. Six different loads are applied at L4 vertebral body. Greg Dakin, et al [18] studied the responses of human symphysis joint of pelvic system under tensile and compressive loads. Their experimental tests revealed that the impacted joint’s symphysis have lesser stiffness than unimpacted ones. Andrew E Anderson, et al [19] studied the mechanical properties of pelvis bone structure. He indicated that the average thickness of ilium bone is around 1.4mm. The experimental technique that was followed during their study is detailed in this article. The schematic they used is shown in figure 2.

Figure 2 Schematic outlining the pelvis testing [19]

2. OBJECTIVE
Advanced manufacturing technologies like 3D printing made the understanding of biomechanics lot easier. 3D printing is being used in medical field in a variety of places like planning cardiac surgery [20], printing bio-models and implants [21], prosthetics [22]–[25], splints [26]–[28] etc. Figures 3 shows the image of one such application that was developed in Mechanical Engineering Department of Swarnandhra College of Engineering and Technology.
The aim of the current work is to understand the response of Femur and Ilium bones. As a part of this work, 3D printing the bio-models of Femur and ilium bones are performed and FE analyses are executed to study the load response.

3. 3D PRINTING BIO MODELS

As indicated earlier, Femur and Ilium bones are 3D printed. Wanhao Duplicator i3 printer is used for this purpose. This has a print volume of 200mm X 200mm X 180mm and can print various materials like PLA, ABS, PVA, PEVA, HIPS using FDM technique [29]. The print speeds vary from 25mm/min to 60mm/min and a minimum layer thickness of 0.1mm. The code for printing is generated using Cura 15.04 software [30]. During printing of models of both the bones, 30mm/min speed is used with a grid infill of 15%. Supports are generated touching the build plate. The build plate is an Aluminium hot plate maintained at 60°C and the printing temperature is 205°C. The nozzle diameter used for printing is 0.4mm. Figures 4 and 5 show the tool paths that are generated for 3D printing femur and ilium. Figures 5 & 6 show the 3D printed models of Femur and Ilium. These bio-models help in understanding the interaction between the bones. It may be noted here that the CAD models are generated from CT Scan images using MIMICS software. The wall thickness of ilium is maintained at 1mm [19] both in CAD model and 3D printed model.
4. FE ANALYSIS OF ILIUM BONE

The cad model of Ilium bone is generated from CT scans. The average thickness of the model is 1mm. Catia is used to prepare the model for FE Analysis. Figure 8 shows the CAD model of Ilium bone. Ilium bone model is then imported into Solidworks for executing FEA. The meshed model is shown in figure 9. 168922 nodes and 84960 solid elements are generated while meshing. During FE Simulations, body weight from 80 to 150 kgs are simulated. Figure 10 shows the principle stress plots for various body weights. The loading and boundary conditions are shown in figure 9 (green is the fixed support while red is the load). It may be noted that the body weight is simulated as the pressure load in the capsule region. Figure 10 show the principle stress plots of ilium bone and Figure 11 gives the principle strain plots of the same. Based on the investigations, it is observed that the peak stresses during static case are within the limits of the ultimate strength of bone material (135MPa in tension and 205MPa in compression). At a weight of 150kg, the max stress, after neglecting degeneracies is found to be 16MPa along the wall of the capsule region. Thus, in normal case, Ilium should not fail.

Figure 6 3D printed Femur

Figure 7 3D Printed Ilium

Figure 8 CAD model of Ilium

Figure 9 Meshed model of Ilium

Figure 10. Principle Stress plots of ilium bone
As per [18], the peak impact load at which the ilium fails is around 100kN and the failure mostly occurs near the symphysis. Thus, this load is simulated. The principle stress plot is shown in figure 12. Large stress values are observed near symphysis which are in agreement to experimental findings in [18]. In other words, a healthy person’s ilium bone can withstand up to an impact load of 100kN.

5. FE ANALYSIS OF FEMUR

The CAD model generated using MIMICS software is imported into ANSYS for executing analysis after some processing using Solidworks. The model generated is a hollow model with an average thickness of 1.4mm. FEA is performed using ANSYS. FE Mesh is generated using Tetrahedral Solid Element with an element size of 3mm. Figure 13 Shows the sectional CAD model. Figure 14 shows the meshed model. 163569 Nodes and 99961 elements are generated. Single leg loading of 100Kgs is simulated. Loading and boundary conditions are shown in figure 15.
Figure 16 shows the principal stress plot for the bone under the specified loading. The blow up in the same figure indicates the stress distribution along the neck region. The Plot shows larger stress along the neck of the Femur bone. It may be noted that the FE analysis is performed for a healthy person. Though the max stress, which is 28MPa is quite less than when compared to that of the max allowable tensile stress, and is having a factor of safety around 2.5, it can be stated that if a fracture is to start, it shall start at the neck region. This is because of the low cross-section in this region. This finding is of importance because, for a person suffering with osteoporosis, and having low BMD, Femur is susceptible for fracture starting in this region. This is agreeing with most common fracture scenarios [31].

![Figure 16 Principal Stress Plot of Femur](image)

6. CONCLUSION
Biomechanics is the study of mechanics of biological systems. This is important so as to understand the mechanical behavior of various body parts so that various types of implants, suits, etc., can be easily designed. 3D printing technology is helping in giving better insight to the interactions of various bones. This can be done quickly through biomodelling and printing the same. In the current work simulations were executed on Ilium and Femur bones so as to understand the response of the same under load. Initially 3D printing is used to bio-model these two bones so as to understand various regions and the way interactions happen between these two. FEA is then performed on these two bones individually so as to understand the behavior of the bones. FEA results of Ilium bone indicated that the Ilium has possible failure region near symphysis and Ilium of a healthy person can take a load of 100kN. FEA results of Femur indicated that the possible failure of Femur under single leg load will be a Femoral Neck Fracture and that too for a healthy person, it can easily support more than 150kg and still at this load has a factor of safety around 2.5.

REFERENCES


