Three-Dimensional Simulation of the Femur Bone Using Finite Element Method

SOMKID AMORNSAMANKUL
Mahidol University
Faculty of Science
Department of Mathematics
Rama VI Road,Bangkok 10400
Centre of Excellence in Mathematics
CHE, Sri Ayutthaya Road, Bangkok 10400
THAILAND
scsam@mahidol.ac.th

BENCHAWAN WIWATANAPATAPHEE
Mahidol University
Faculty of Science
Department of Mathematics
Rama VI Road,Bangkok 10400
Centre of Excellence in Mathematics
CHE, Sri Ayutthaya Road, Bangkok 10400
THAILAND
scbww@mahidol.ac.th

KAMONCHAT KAORAPAPONG
Mahidol University
Faculty of Science
Department of Mathematics
Rama VI Road,Bangkok 10400
THAILAND
g5237091@mahidol.ac.th

Abstract: In this paper, a mathematical model is developed to simulate three-dimensional femur bone, taking into account the stress distribution and total displacement during horizontal walking. The principles of continuum mechanics, the field equations governing the displacement and stress fields in the femur bone including the stress equilibrium equations are used in the model. Realistic domain is created using CT scan data. Three different cases of static loads and three boundary conditions are used in the simulation. The finite element method is utilized to determine the total displacement and the Von Misses stress. The influences of human weight during horizontal walking are investigated. This model provides useful information for surgeons involved with femur surgeries.

Key–Words: Femur bone, Finite element method, Simulation, Mathematical modeling, Total displacement.

1 Introduction

For several centuries, human has tried to understand all parts of the human body. Biologists and medical staff have done several experiments to gain deeper knowledge about human's organs. It took a long time and a lot of work in order to discover such knowledge. After the computer has emerged, mathematicians have applied mathematical models to explain how a human's organ such as brain, heart, blood, lung and bone works. These studies are very useful for orthopedic surgeons.

One of the most important organs in the human body is the femur. For example, the distribution of stress using experimental analysis across the neck of the femur has been investigated [8]. In vitro experiments were also conducted to analyse the distribution of stress across the neck of the femur [7]. However, the shear stress distribution was not reproduced. Finite element models were developed for normal and osteoarthritic femur using two-dimensional plane [2]. It was found that regions of extremely low stresses were observed at the outer layers of the femur head. Some researchers developed three-dimensional mathematical models to investigate the geometry of femur and stress [1, 3]. Krauze [5] also studied a numerical simulation for stresses and displacement on femur in a living and a dead phase. The influence of mechanical properties of bone tissues were also investigated in their studies. However, the realistic geometry of femur bones is still a subject for active research.

This paper aims to construct a complete three-dimensional femur bone from CT scan data. The Mimics commercial software is used to create 3D models and smooth the surface of the domain. The finite element method is applied to find the stress distribution and total displacement. The rest of the paper is organized as follow. Section 2 describes the governing equations in the mathematical model. The do-

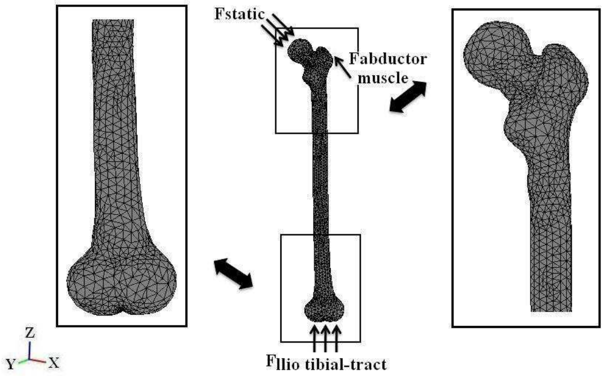

Figure 1: Three-dimensional geometry of fumer bone and its finite element mesh.

main, boundary conditions and three different cases of static load are also shown in Section 2. The results are consequently shown in Section 3. Finally, Section 4 contains the concluding remarks and suggestion of future works.

2 Mathematical Model

Three-dimensional model of the right femur was constructed using CT scan data of a human femur. Fig. 1 shows the complete geometry of the right femur bone of length 50 cm with forces acting on the head and the end of femur. Mesh of the femur bone geometry consists of 17,495 tetrahedral elements and 87,288 degrees of freedom. The bone is assumed to be a Von Mises elasto-plastic material. From the principles of continuum mechanics, the field equations governing the displacement and stress fields in the femur bone including the stress equilibrium equations are

$$\begin{aligned} \sigma_{ij,j} + f_i &= 0 \quad (i = 1, 2, 3) && (1) \\ \varepsilon_{ij}(\vec{u}) &= \frac{1}{2}(u_{i,j} + u_{j,i}), && (2) \\ \sigma_{i,j} &= (C_{ep})_{ijrs}\varepsilon_{rs}, && (3) \end{aligned}$$

where σ and ε denote respectively the stress tensor and the strain tensor, $\vec{u}$ is displacement, f_i is body force and $(C_{ep})_{ijrs}$ is a tensor of elastic constants, or a moduli which are independent of stress or strain.

Table 1 shows the parameters used in the numerical simulation. In order to simulate the stress field corresponding to the walking activity, we impose three boundary conditions based on the human's weight as follows [4]:

Table 1: Values of the parameters used in computational region [6].

$Parameters$	Femur	Units
Density (ρ)	2000	kg/m^3
Young's modulus (E)	2.13	GPA
Poisson's ration (ν)	0.3	–
Yield strength	1.48$\times$ 10^8	Pa

(I). $F_{dynamic}$: F_x= 234 N, F_y = 385 N, F_z = -1652 N

$F_{abdductor}$: F_x = 0 N, F_y= 0.8 N, F_z = -1.937 N

$F_{Illiotibial-tract}$: F_x = 0 N, F_y = 0 N, F_z = 350 N

(II). $F_{dynamic}$: F_x= 334.8 N, F_y = 550 N, F_z = -2360 N

$F_{abdductor}$: F_x = 0 N, F_y= 1.142 N, F_z = -2.8 N

$F_{Illiotibial-tract}$: F_x = 0 N, F_y = 0 N, F_z = 500 N

(III). $F_{dynamic}$: F_x= 669.6 N, F_y = 1100 N, F_z = -4720 N

$F_{abdductor}$: F_x = 0 N, F_y= 2.285 N, F_z = -5.6 N

$F_{Illiotibial-tract}$: F_x = 0 N, F_y = 0 N, F_z = 1000 N

The loads (I), (II) and (III) represent terminal stance during horizontal walking in which each person has the weight of 70, 100 and 200 kg, respectively.

3 Results and discussion

In this study, the effect of the human's weight on the total displacement and von Misess stress during horizontal walking have been investigated. In Fig. 2 the total displacement of the loading approximates the peak gait load for a 70 kg person during horizontal walking. The results show that high displacement ap-

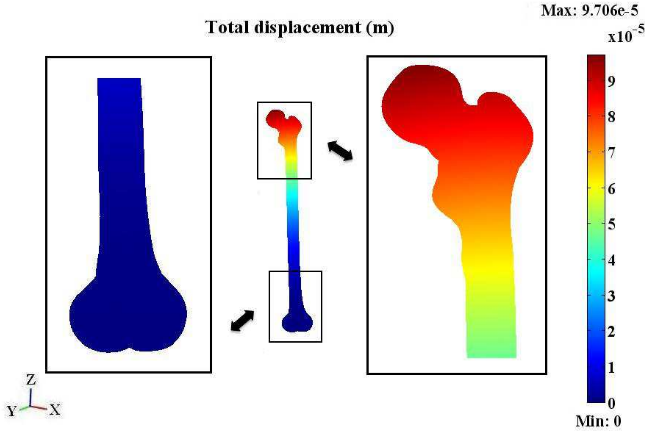

Figure 2: Total displacement of the loading approximates the peak gait load for a 70 kg person during horizontal walking.

pears at the head of the femur bone and the lower displacement occurs at the end of the femur. The total displacement for different human weights are compared in Fig. 3. It is found that higher weight provides higher total displacement and lower weight provides lower total displacement.

Front view and back view of Von Misess stress are shown in Fig. 4. The results indicate that higher Von Misess Stress is located at the front view of the end of femur. Additionally, the Von Misess stress indirectly affects the lateral femur bone.

4 Conclusions

The forces acting on the head and the end of the femur bone have been studied numerically using a three-dimensional mathematical model and a numerical technique based on the finite element method. The numerical investigation shows that high displacement occurs at the head of the femur whereas the lower displacement occurs at the end of the femur. The results also show that higher weight provides higher total displacement. Moreover, it is found that the Von Misess stress affects the lateral femur. It should be noted that this paper focuses only on the normal femur. In the future, we will apply real geometry to simulate the cemented hip replacement in order to obtain useful information and better understanding which could be of great help to orthopaedic surgeons.

Acknowledgements: The research is (partially) supported by the Centre of Excellence in Mathematics, the Commission on Higher Education, Thailand.

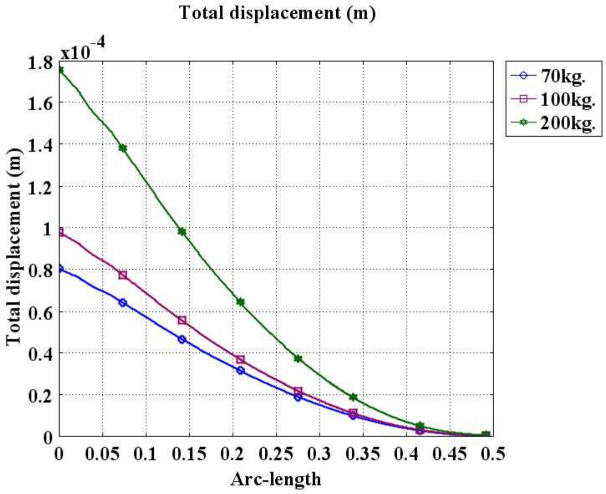

Figure 3: Profile of total displacement along the axial of femur bone where the solid-circled line corresponds to the load of 70 kg, the solid-squared line corresponds to the load of 100 kg, and the solid-starred line corresponds to the load of 200 kg.

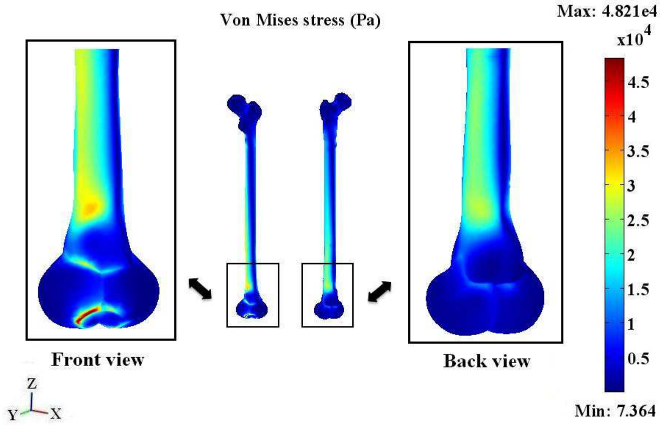

Figure 4: Von Misess stress of the loading approximates the peak gait load for a 70 kg person during horizontal walking.

References:

[1] T.R. Deshmukh, A.M. Kuthe, D.S. Ingole, and S.B. Thakre. Prediction of femur bone geometry using anthropometric data of india population: A numerical approach. *International Journal of Medical Science*, 10(1):12–18, 2010.

[2] A.H. Elkholy, D.N. Ghista, F.S. D'Souza, and M.S. Kutty. Stress analysis of normal and osteoarthritic femur using finite element analysis. *International Journal of Computer Applications in Technology*, 22(4):205–211, 2005.

[3] Royi Fedida, Zohar Yosibash, Charles Milgrom, and Leo Joskowicz. Femur mechanical simulation using high-order fe analysis with continuous mechanical properties. In H. Rodrigues et al., editor, *II International Conference on Computational Bioengineering*, Lisbon, Portugal, September 14-16 2005.

[4] Russell K. Hobbie and Bradley J. Roth. *Intermediate Physics for Medicine and Biology*. Springer, fourth edition, 2007.

[5] A. Krauze, M. Kaczmarek, and J. Marciniak. Numerical analysis of femur in living and dead phase. *Journal of Achievements in Materials and Manufacturing Engineering*, 26(2), February 2008.

[6] Srimongkol S. and Wiwatanapataphee B. *Simulations in Cemented hip replacement*. PhD thesis, Faculty of Science, Mahidol University, 2008.

[7] S. Valliappan, N.L. Svensson, and R.D. Wood. Three dimensional stress analysis of the human femur. *Computers in Biology and Medicine*, 7(4):253–264, 1977.

[8] J.F. Williams and N.L. Svensson. An experimental stress analysis of the neck of the femur. *Medical and Biological Engineering and Computing*, 9:479–493, 1971.