International Journal of Applied Engineering Research
ISSN 0973-4562 Volume 10, Number 9 (2015) pp. 22875-22888

http://www.ripublication.com

Finite Element Analysis of Femur Bone Under Different Loading Conditions For Healthcare System

R. Vijayakumar M. Madheswaran
Department of Electronics and Communication Engineering,
Mahendra Engineering College, Mallasamudram -637 503, TN, India.
rvijay.nethra@gmail.com, madheswaran.dr@gmail.com

Abstract

The 3D model of femur bone is developed for calculating the stress distribution and total displacement during horizontal walking. Different boundary and static load conditions are applied in the simulation. The Total displacement and Von Mises Stress are measured using Finite Element Method (FEM). It is found that higher total displacement occurred when higher weights loaded also femur head can defend against larger load which may occur under sudden twist or tumble cases due to larger material strength and size of it.

Keywords: Femur bone, Finite Element Method (FEM), Three dimensional simulation, Dynamic conditions, Implant in the femoral canal

1. Introduction

Femur bone is an essential part of human body which provides support to the human body in standing, sitting and walking. It is complex in shape and has different composition. The thickness of femur bone is about 4-8mm and length is between 260-293mm for research has been carried out on femur bone fracture using mathematical modeling and different software's like ANSYS,PRO/Engineer, MIMICS etc.. Usually MRI, CT scan and advanced imaging techniques were employed to prepare the model for simulation purposes which is expensive. Finite element method is one of the best ways for linear, nonlinear and couple field analysis of biological parts.

However, much effort was devoted to consider the load condition associated with normal human activities and very few studies considered the loading mode of sudden twist or tumble. The simplified mechanical model and a two-dimensional FEM model are established based on the numerical result the deformations of femur and stress distributions are obtained. The femur bone is the most proximal bone of the leg in vertebrates capable of walking or jumping. The knee joint is created by femur at its bottom portion meshes with the tibia bone. To create the hip joint the femur meshes with the acetabulum are at its top end [2]. The femur is most important for bearing the largest percentage of body weight during normal weight-bearing activities.

Stulpner M. A, Reddy et al., [2] have performed the Finite Element Analysis (FEA) with the help of a three dimensional model of real proximal human femur bone and its behavior was analyzed by solving the governing equation using FEM method under physiological load conditions. Enrico Schileo and Fulvia Taddei [3] developed the strain based failure criterion based on numerical study and presented the failure patterns of bones with the help of accurately predicted strain by using subject specific finite element models. The identified failure loads are applied to the model and calculated the risk of fracture and compared with the existing results.

Ninja P. Oess, Weisse, et al., [4] presented a sensor signal detection system, to determine the stress and strain in the plate subject to tensile load and compressive load, using nonlinear Finite Element Analysis (FEA) performed with ABAQUS/Standard V. 6.5.4. Their result shows that the inner surface of the plate at the screw holes has the maximum stress and maximum strains in the plate. ZHAO June-hai and MA Shu-fang [5] are focused the loading mode and stress distribution under bending and compression conditions which may occur sprains or tumble. Their investigation was based on FEM. Tomasz Topolinski, Adam Mazurkiewicz [6] evaluated the correlation between bone strength using the values of volume, fracture dimension and bone mineral density. The compression force F was calculated using finite element analysis method.

X.G. Mao, J.H. Zhao, G.Q. Zhang et al [7], W.Q. Zhang et al., [8] Z.X. Qi et al., [9] are found that the behavior of femur is necessary to get a clear idea about femur fracture and provide better guidance to the artificial femur replacement. To examine the loading mode and stress distribution more work has been conceded. J. Williams, N. Svensson [10] and S. Valliappan, R. Wood et al.,[11] conducted experiments to analyze the distribution of stress across the neck of the femur; however, the shear stress distribution was not satisfied. A. Elkholy and D. Ghista et al., [12] developed the finite element models for normal femur. By using this model they analyzed the stress of normal femur with osteo arthritic femur.

Three dimensional mathematical femur bone ANSYS model was developed by T. Deshmukh, A. Kuthe et al., [13] and R. Fedida, Z. Yosibash et al., [14] to investigate the geometry of femur and stress. Femur stress and displacement in a living and a non living phase was demonstrated using numerical simulation by Krauze et al., [15]. M. Numerical simulation of the femur bone was performed by Pawlikowski [16] and found that the boundary conditions and different types of load has maximum influence in generating fracture in femur bone. N. G. Bizdoaca et al.,[17] and V. Volpe et al.,[18] investigated about the bone fracture using in X-ray based technique and they suggested the modular adaptive implants for fractured bone.

2. Methodology

Finite Element Analysis of femur bone under physiological condition is essential for the understanding of failure mechanisms and providing guidance for the design and operation of femur replacement. To create a CAD model by using PRO/Engineer, Solid Edge, MIMICS, etc from the obtained the patient's anatomical data is either in radiograph or CT-MRI form. Stresses and deformation in different activities were analyzed using Finite Element Analysis. Human bone is highly heterogeneous and nonlinear in nature, so it is difficult to assign material properties along each direction of bone model. In biomechanics study, material can be assign in two ways, either in Mimics or in finite element module. In ANSYS directly dispersed as properties of material. Table 1 shows the femur bone properties assigned in ANSYS.

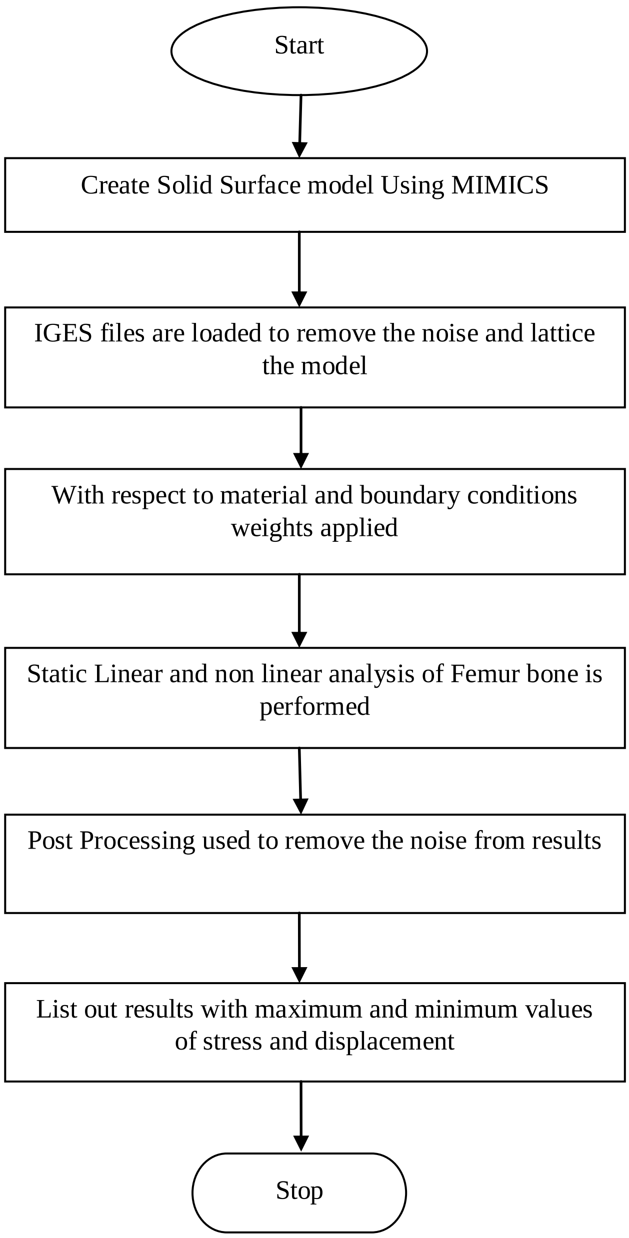

Figure 1: Flow graph for Femur Analysis Method

Based on anthropometric measurements two 3D finite element models are developed in the order of 1069 nodes and 508 elements. A proximal side of the thighbone head has force of 2500N arises due to thighbone loaded. The center of distal epiphysis and the proximal side femur head of center both are interconnected at the force of vector happened in axis (Z axis). We used the support of distal parts of both condyluses that permitted free displacement in orthogonal plane (denoted as X, Y axis) to the force vector (denoted as Z axis).

Table 1: Material Properties assigned to femur

S. No.	Item	Elastic Constant (E), MPa	Poission’s Ratio [υ]
1	Head	900	0.29
2	Neck	620	0.29
3	Shaft	17000 to 14000	0.29
4	Bone Marrow	100	0.29
5	Cortical bone	17650	0.25
6	Cancellous bone	355	0.25
7	Trochanter	260	0.30
8	Implants (steel)	$2x10^5$	0.32
9	TiAl6V4	$1.15x10^5$	0.34
10	Titanium Grade 1	$1.03x10^3$	0.34

The field equations leading to displacement and the stress fields in the femur bone with the continuum mechanics principles, which include the stress equilibrium equations as given below:

$$\sigma_{ij,j} + f_i = 0 \,(i = 1,2,3) \qquad \text{..........} \qquad (1)$$

$$\varepsilon_{ij}(\vec{u}) = \frac{1}{2}\left(u_{i,j} + u_{j,i}\right) \qquad \text{..........} \qquad (2)$$

$$\sigma_{i,j} = \left(C_{ep}\right)ijrs\varepsilon_{rs} \qquad \text{..........} \qquad (3)$$

The stress tensor and the strain tensor are represented as σ and ε respectively, displacement is $\vec{u}$, body force is f_i and tensor of elastic constants $\left(C_{ep}\right)ijrs$, or a modulus which are independent of stress or strain.

The loads (I), (II) and (III) represent terminal stance during horizontal walking in which each person has the weight of 80, 110 and 160 *kg* respectively. The displacement is reserved on the distal epiphysis (the base). To create the force acting on the entity related to patient’s activity for that load is imposed on the implant head external point. The implant load (I) anticipated the load for an 80 kg person during normal walk [20] and the implant load (II) anticipated the terminal stance during horizontal walking [21].

Table 2: Forces Used For Numerical Simulation of Implant

Implant load	Force (N)	$F_{dynamic}$	$F_{abddductor}$	$F_{lliotibial\text{-}tract}$
I	F_x	237	0	0
	F_y	389	0.85	0
	F_z	-1655	-1.939	352
II	F_x	335.5	0	0
	F_y	552	1.145	0
	F_z	-2361	-2.9	505
III	F_x	669.9	0	0
	F_y	1101	2.295	0
	F_z	-4720.5	-5.65	1010

3. Results and Discussion

ANSYS 14.0 (Workbench) is used to analyse the static structure. An intact human femur of three-dimensional solid finite element model is constructed using linear elastic, isotropic and homogeneous material properties. The femur bone is verified before analyzing the fractured bone. This task proceeded as per which contained stress values at medial and lateral side of human femur.

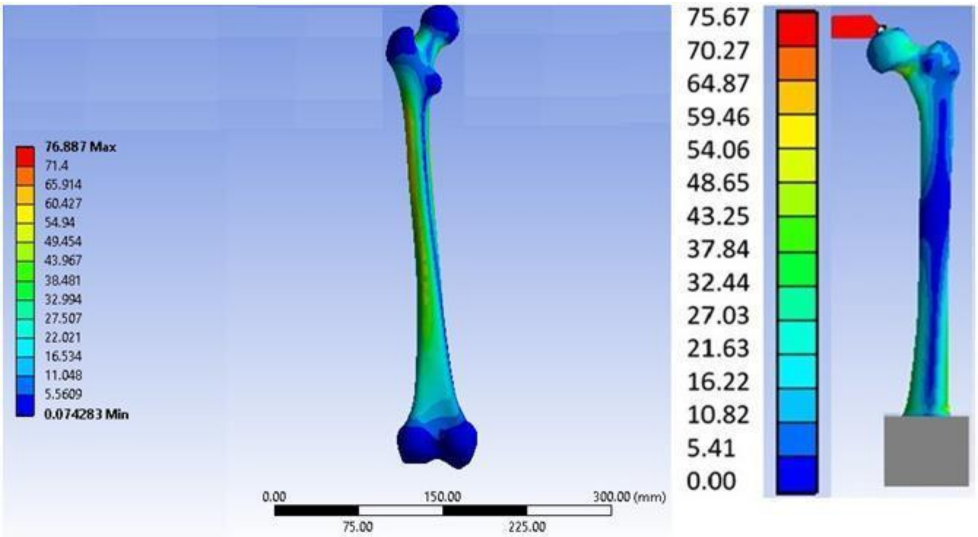

Figure 2: Femur Model

A femur bone of height 415 mm was taken and the stress values in 15 points were analyzed and compared with the earlier model as shown in Figure 2. The stress results found slight differences between the earlier femur and our femur model. These differences were caused by different geometry structure of the femur and the finite element quality of the femur model depends on the hardware parameters of used computers.

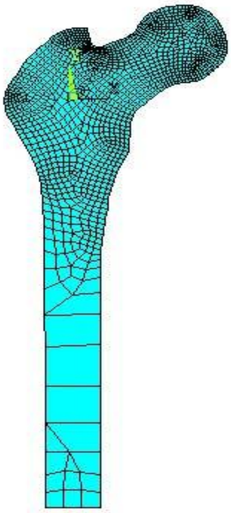

Figure 3: Finite element model of femur

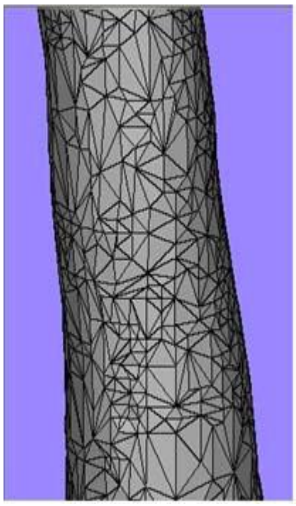

Figure 4: Surface mesh

The numerical simulation is carried out based on the commercial finite element code ANSYS 8.0. A two-dimensional numerical model as shown in Figure 3 is set up based on the real dimension of human femur. The plate 42 element is used for the model. There are a total of 1494 elements and 1599 nodes in the model.

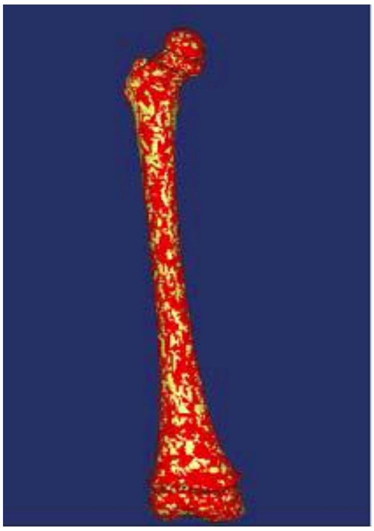

Figure 5: Volumetric mesh of femur bone

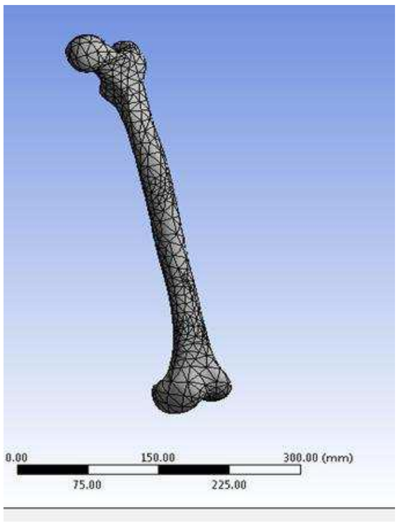

Figure 6: Mesh generation on femur

The three dimensional model of femur bone with volumetric mesh is used for FEA analysis. Figure 4 shows surface mesh of femur bone and Figure 5 shows volumetric mesh of femur bone in mimics. In the first domain, the effects of the human's weight on the total displacement and Von Mises stress during horizontal walking have been investigated. Analogous to the real experiment, we permitted a rotation around Z axis in our analyses. For the field as shown in Figure 6, enforce boundary conditions based on the human's weight [19].

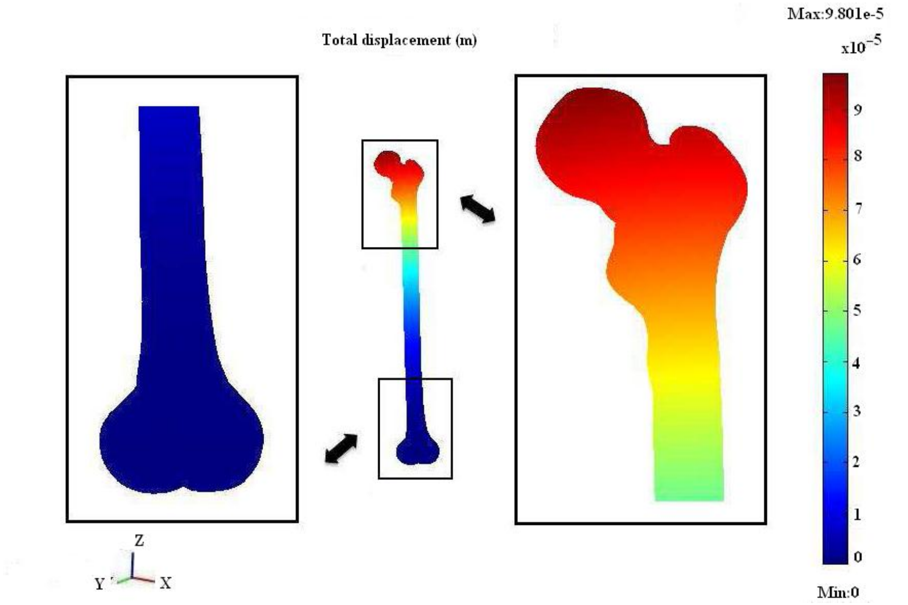

Figure 7: Total displacement of femur bone for 80 kg person in horizontal walking

For 80kg person during horizontal walking has maximum total displacement occurs at top of the femur head and minimum at bottom shown in the Figure 7. The maximum displacement is 9.801×10^{-5} and minimum displacement is 6.150×10^{-5} approximately.

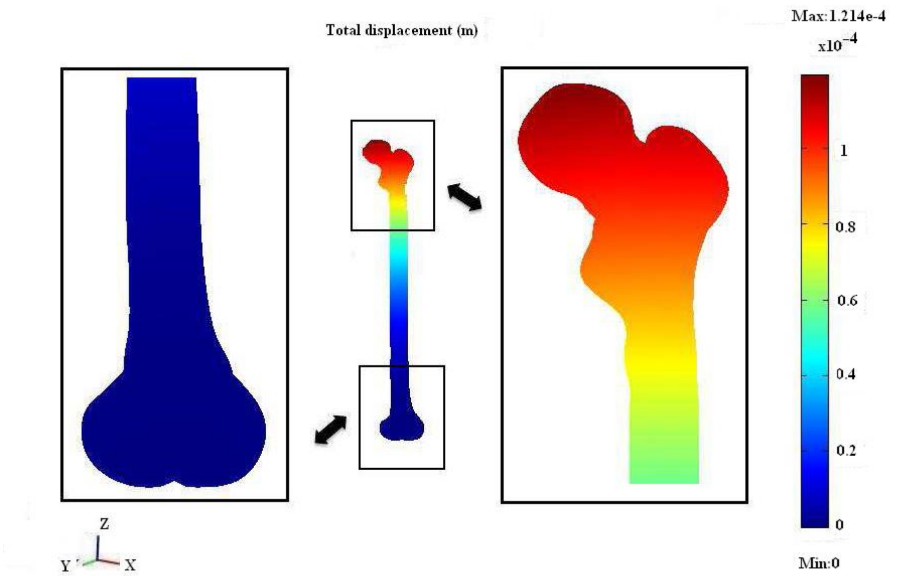

Figure 8: Total displacement of femur bone for 110 kg person in horizontal walking

The Figure 8 results shows that maximum displacement is 1.214×10^{-4} and minimum displacement is 0.650×10^{-4} approximately for 110kg person in horizontal walking. Compare with the 80kg person total displacement 110kg person has high displacement appears at the head of the femur bone and at the end of the femur bone.

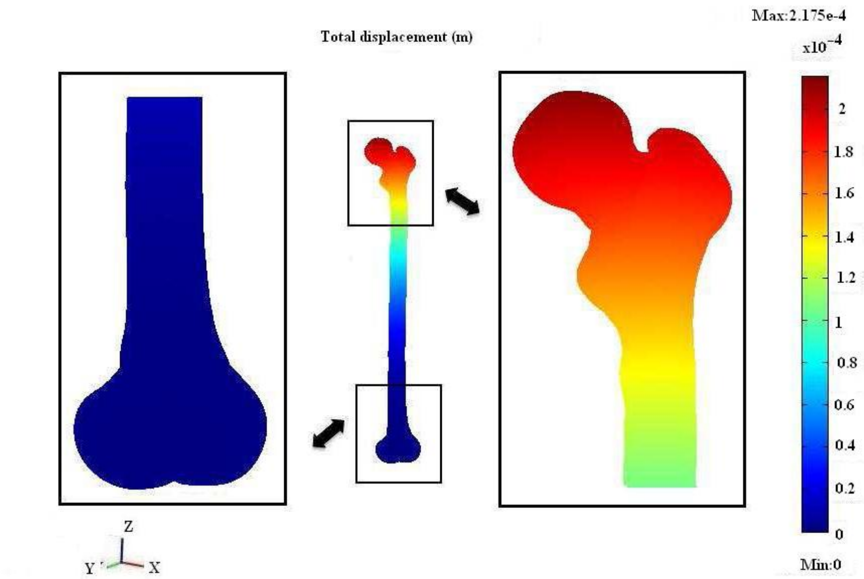

Figure 9: Total displacement of femur bone for 160 kg person in horizontal walking

The Figure 9 result shows that maximum displacement is 2.175×10^{-4} and minimum displacement is 1.250×10^{-4} approximately for 160kg person in horizontal walking. Compare with the 80kg, 110kg persons total displacement 160kg person has more displacement appears at the head of the femur bone and at the end of the femur bone. It is found that higher weight provides higher total displacement and lower weight provides lower total displacement.

Front view and back view of Von Mises stress are shown in Figure 10-12. The results indicate that higher Von Mises Stress is located at the front view of the end of femur.

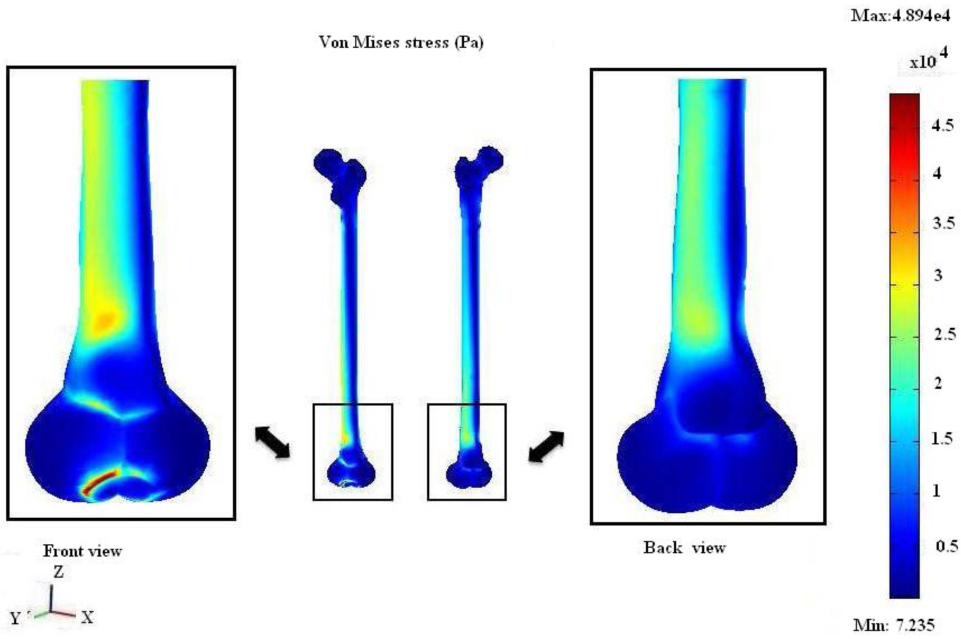

Figure 10: Von Mises stress of femur bone for 80 kg person in horizontal walking

Additionally, the Von Mises stress indirectly affects the lateral femur bone. In this part of study, we consider the total displacement and von Mises stress for a 80 *kg* person during normal walk (I) and terminal stance horizontal walking (II).

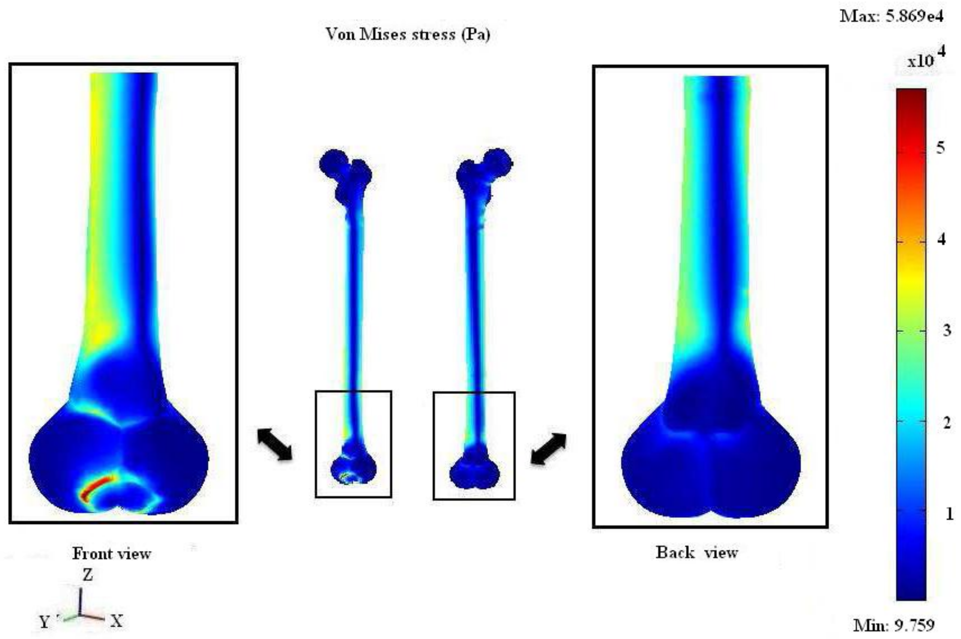

Figure 11: Von Mises stress of femur bone for 110 kg person in horizontal walking

For the implant load (I) and implant load (II), the force acting on the object corresponding to patient's activity imposed at the head implant external point is simulated. The displacement is reserved on the distal epiphysis (the base).

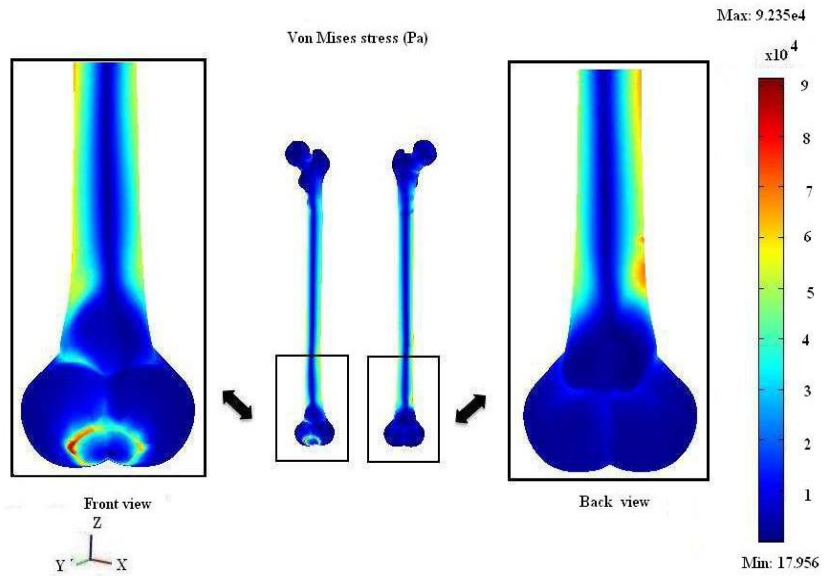

Figure 12: Von Mises stress of femur bone for 160 kg person in horizontal walking

For the implant load (I), we obtain that the total displacement of implant and femoral canal in artificial femur bone based on real domain for a 80 *kg* person during normal walk is $4.097x10^{-4}$ m.

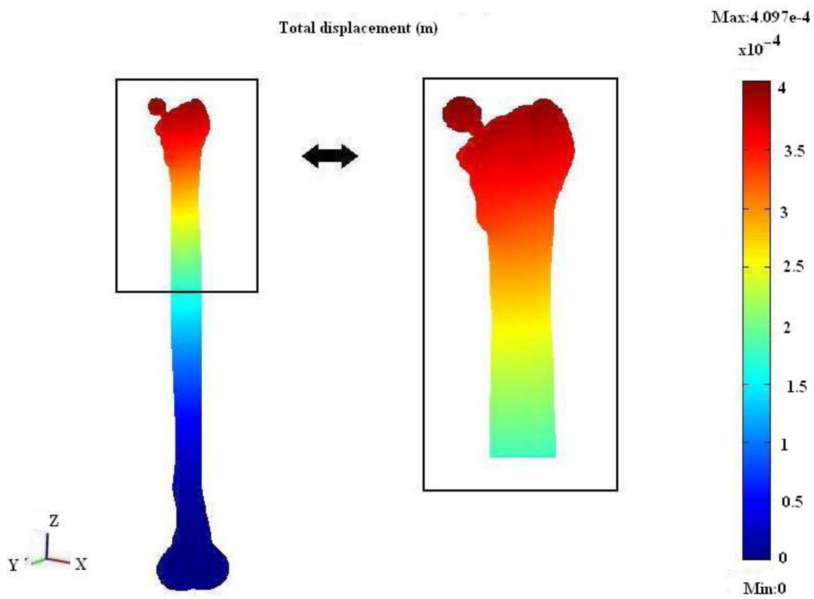

Figure 13: Femur bone total displacement during normal walking for 80 kg person

Figure 13 presents the total displacement of implant and femoral canal in artificial femur bone based on real domain for a 80 *kg* person during normal walking (I). The result shows that high displacement appears at the head of implant and lower displacement occurs at the end of the femur. It indicates that the total displacement along the axial of artificial femur bone is larger than the along the axial of femoral canal.

Table 3: Total displacement and Von Mises stress for different weights of human

Weight of person (kg)	Total Displacement (m)		Von Mises stress (Pa)	
	Minimum	Maximum	Minimum	Maximum
80	$0.615\text{x}10^{-4}$	$0.980\text{x}10^{-4}$	7.235	$48.94\text{x}10^{3}$
110	$0.650\text{x}10^{-4}$	$1.214\text{x}10^{-4}$	9.759	$58.69\text{x}10^{3}$
160	$1.250\text{x}10^{-4}$	$2.175\text{x}10^{-4}$	17.956	$92.35\text{x}10^{3}$

With this table we conclude that minimum displacement and Von Mises stress occurs due to low weight while maximum value is obtained when the human weight get increased to high.

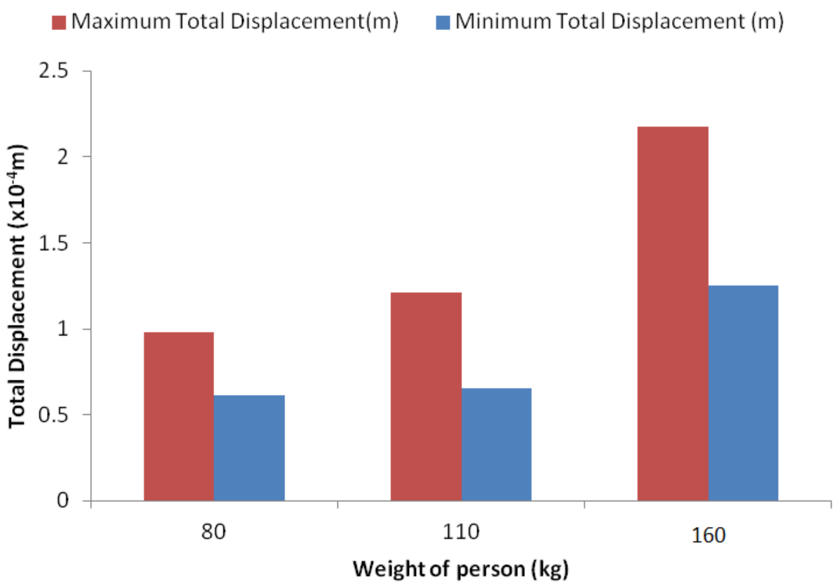

Figure 14: Total displacement for different weights of human

Total displacement of implant and femoral canal in artificial femur bone based on real domain for 80 kg, 110 kg and 160 kg persons during normal walking (I) shown in Figure 14. With analysis of results the higher weights produce higher displacement at the femur head compare to lower ends of femur.

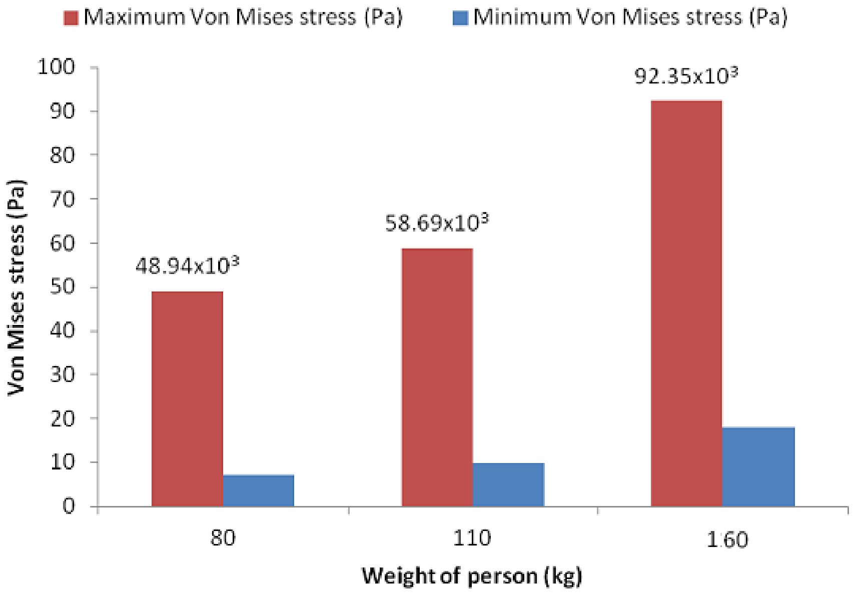

Figure 15: Von Mises Stress for different weights of human

Total Von Mises Stress of implant and femoral canal in artificial femur bone based on real domain for 80 kg, 110 kg and 160 kg persons during normal walking (I) shown in Figure 15. Stress concentration minimum at the weak parts of femur neck and the upper ends of femur shaft. For this reason, during fracture treatment and physical exercise shielding these parts is to be immense substance.

4. Conclusion

Three-dimensional statistical model and a geometric technique based on the finite element process are used to study the forces acting on the head and the end of the femur bone and implant in the femoral canal. The mathematical analysis shows that at the head of the femur has high displacement while at the end of the femur has lower displacement occurred. The results show that if the weight is higher the total displacement is also higher. The middle and lower ends of femur neck has evident stress concentration is identified.

5. References

[1] Tony M. Keaveny, Elise F. Morgan and Oscar C. Yeh “Standard Handbook of Biomedical Engineering and Design” The McGraw-Hill publications2004; pp 8.1-8.23.

[2] Stulpner M.A,Reddy B.D,Starke G.R and Spirakis A, “A Three – Dimensional Finite Analysis Of Adaptive Remodelling in The Proximal Femur”, Journal of Biomechanics, 1997; Vol. 30.N0. 10, pp 1063-1066.

[3] Enrico Schileo,FulviaTaddei, “Subject-specific finite element models implementing a maximum principal strain criterion are able to estimate failure risk and fracture location on human femurs tested in vitro” Journal of Biomechanics 41, 2008; pp. 356-367.

[4] Ninja P.Oess, Bernhard.Weisse, and Bradley J, “Magnetoelastic Strain Sensor for Optimized Assessment of Bone Fracture Fixation” IEEE SENSORS Journal, August 2009; Vol. 9, NO.8.

[5] ZHAO Jun-hai,MA Shu-fang, “Finite Element Analysis of femur stress under bending moment and compression load” Biomedical Engineering and Informatics2009; BMEI '09. 2nd International Conference on 10.1109/BMEI.2009.5305316.

[6] Tomasz Topolinski,Adam Mazurkiewicz, “Micro architecture Parameters Describe Bone Structure and Its Strength Better than BMD” The Scientific World Journal, Vol. 2012,Article ID 502781,7 pages doi:10.1100/2012/502781.

[7] X.G. Mao, J.H. Zhao, G.Q. Zhang et al. “The stress analysis of the upper femur”, Chinese Journal of Clinical Anatomy, 1996; 14(3): pp. 234-236.

[8] X.G. Mao, J.H. Zhao, W.G. Zhao et al. “The Photoelastic Study of the Force Condition of the Upper Femur”, Journal of Xi’an Medical University, 1996; 17(3): pp. 330-332.

[9] Z.X. Qi, D.X. Zhang, Q.S. Wei, et al. “Study of the Distribution of Stress in Human Femur under Loading”, Journal of Practical Orthopedics, 2003; 9(3): pp. 230-233

[10] J. Williams and N. Svensson, “An experimental stress analysis of the neck of the femur,” Medical and Biological Engineering and Computing, 1971; vol. 9, pp. 479–493.

[11] S. Valliappan, N. Svensson, and R. Wood, “Three dimensional stress analysis of the human femur,” Computers in Biology and Medicine, 1977; vol. 7, no. 4, pp. 253–264.

[12] A. Elkholy, D. Ghista, F. D’Souza, and M. Kutty, “Stress analysis of normal and osteoarthritic femur using finite element analysis,” International Journal of Computer Applications in Technology, 2005; vol. 22, no. 4, pp. 205–211.

[13] T. Deshmukh, A. Kuthe, D. Ingole, and S. Thakre, “Prediction of femur bone geometry using anthropometric data of india population: A numerical approach,” International Journal of Medical Science, 2010; vol. 10, no. 1, pp. 12–18.

[14] R. Fedida, Z. Yosibash, C. Milgrom, and L. Joskowicz, “Femur mechanical simulation using high-order FE analysis with continuous mechanical properties,” in II International Conference on Computational Bioengineering, H. R. et al., Ed., Lisbon, Portugal, September 2005;pp 14-16.

[15] A. Krauze, M. Kaczmarek, and J. Marciniak, “Numerical analysis of femur in living and dead phase,” Journal of Achievements in Materials and Manufacturing Engineering, February 2008; vol. 26, no. 2.

[16] M. Pawlikowski, "Computer simulation of bone adaptation process under various load cases," in Proceedings of the 2006 WSEAS international conference on Cellular and Molecular Biology, Biophysics and Bioengineering. Athens, Greece: WSEAS, July 2006; pp. 117–122.

[17] N. G. Bizdoaca, D. Tarnita, D. Popa, and E. Bizdoaca, "Shape memory alloy based modular adaptive ortophedic impants," in Proceedings of the 1st WSEAS International Conference on Biomedical Electronics and Biomedical Informatics. Rhodes, Greece: WSEAS, August 2008; pp. 188–195.

[18] V. Volpe, C. Miraglia, L. Esposito, and M. Fraldi, "X-ray based technique for estimating bone fracture risk," in Proceedings of the 2nd WSEAS International Conference on Biomedical Electronics and Biomedical Informatics. Moscow, Russia: WSEAS, August 2009; pp. 244–246.

[19] R. K. Hobbie and B. J. Roth, Intermediate Physics for Medicine and Biology, 4th ed. Springer, 2007.

[20] H. El-Sheikh, B. MacDonald, and M. Hashmi, "Material selection in the design of the femoral component of cemented total hip replacement," J Mater Process Technol, 2002; vol. 122, pp. 309–317.

[21] H. Katoozian and D. Davy, "Effects of loading conditions and objective function on the three-dimensional shape optimization of femoral components of hip endoprostheses," Med Eng Phys, 2000; vol. 22, pp. 243–251.

[22] Somkid Amornsamankul, Kamonchat Kaorapapong, Benchawan Wiwatanapataphee, "Three-Dimensional Simulation of Femur Bone and Implant in Femoral Canal using Finite Element Method," International Journal Of Mathematics And Computers In Simulation, 2010; Issue 4, Volume 4.

[23] Raji Nareliya and Veerendra Kumar, "Biomechanical analysis of human femur bone," International Journal of Engineering Science and Technology (IJEST), Vol. 3 No. 4 Apr 2011; pp.3090-3094.

[24] D.Amalraju and Dr.A.K.Shaik Dawood, "Mechanical Strength Evaluation Analysis of Stainless Steel and Titanium Locking Plate for Femur Bone Fracture," Engineering Science and Technology: An International Journal (ESTIJ), ISSN: 2250-3498, June 2012; Vol.2, No. 3.

[25] Ashwani Kumara, Himanshu Jaiswala,Tarun Garga, Pravin P Patil, "Free Vibration Modes Analysis of Femur Bone Fracture Using Varying Boundary Conditions Based on FEA", Procedia Materials Science 6 (2014) 1593 – 1599.