International Conference on Mechanical, Industrial and Materials Engineering 2017 (ICMIME2017)
28-30 December, 2017, RUET, Rajshahi, Bangladesh.

Paper ID: AM-118

Modeling and Analysis of Elastic Fields of Human Femur

Parvej, M S., A.M. Afsar, Rafi, T H., Ahmed, S., S M Shayak Ibna Faruqui.

[1]Department of Mechanical Engineering
Bangladesh University of Engineering and Technology, Dhaka, Bangladesh
E-mail: kowshikparvej@yahool.com

Abstract

Elastic field analysis of femur subjected to body weight in both static and cyclic condition was performed in this study. The human femur was fabricated by casting process and taking the cross sections of the femur at different height. A 3-D solid model of human femur was developed in SolidWorks by using the geometry of cross sections at different locations of the fabricated femur. The 3-D model was analyzed by ANSYS to evaluate the stress, strain, and deformation for identifying the critical sections of femur. The critical zones, in terms of total deformation, normal stress and shear stress for both static and cyclic load were identified.

Keywords: Femur, Elastic Field, Modeling, Finite Element Analysis, Casting.

1. Introduction

Femur, also known as thigh bone, is the strongest bone of the human body, bears body weight in lower extremity. It is the largest bone of the body contributing on an average 26.74% of a person's height. The practice of involving Finite Element Method (FEM), in the biomechanical analyses dates way back to 1972[1] that was an evaluation of basic stresses of bones. Taking the better functionality of prosthesis and corresponding correct material choice as a challenge. It was followed by the further analysis on the prosthesis and intrinsic Fracture Mechanics. In this quest, the axisymmetric finite element model developed by Hayes et al. in 1977[2], picturing the Lateral Tibial Plateau, was remarkable as an early development in this sphere. Again, research on the effect of stress distribution on fibula on lower extremity under the body weight loading and torsion moment is carried by Özkan & Kişioğlu[3]. Similar types of researches were accomplished on the other parts of the lower limb where Ozen[4] et al. conducted CT scan and MR to design and analyze stresses on the normal foot-ankle. Popa et al. (2006) presented the method and steps to model a virtual bone [5]. The model was prepared with the help of the CAD software and it was attached to the other bones. In their study, all the important steps explaining all the features required for the modeling of a human femur were explained in detail that can be utilized for the model development and can further be used for elastic field analysis of human femur. Francis and Kumar (2012) performed the three dimensional finite element modeling using Computed Tomography (CT) data that accurately predicts information about bone morphology and tissue density [6]. A 3-D finite element model was created for an intact femur & a synthetic femur implanted with cementless prosthesis for investigating proximal load transfer under two loading conditions by using a couple of experiments and FE analysis [7]. The approach was used to investigate a press fitted and a fully bonded bone prosthesis structure to identify the stem bone behavior for both interface conditions & their implications for proximal bone load transfer. As to our knowledge, although some significant research works have been carried out on the lower limb bones separately, works on the combined model of human lower limb are not that much ample. Besides, the medical imaging techniques such as CT scan or MR procedures have limitations to capture the image of the medullar cavity and the bone cortex properly. In continuation of the previous research works [8], this investigation aims at developing a 3-D solid model of the human lower limb associated with medullar cavity and to use this model to carry out finite element analysis in order to understand its behavior under various loading conditions.

2. Modeling and Simulation

For modeling purpose, original human femur was used as pattern and the casting procedure was solely a sand casting. Figure 1 shows the mold that was formed using these patterns. Shrinkage, taper and machining allowances were considered preparing the mold. Aluminum was used as casting material. Firstly, aluminum was melted and then poured into an enclosed mold. Then, it was cooled naturally for about 24 hours and the sand mold was removed. After cleaning and machining, the final model was produced as showed in Fig. 2.

Fig. 1 Mold for Femur **Fig. 2** Casting Model of Femur

The casting models were sectioned in this stage with power saw that has been depicted in Fig. 3. It is to be mentioned that no type of medullar cavity was done in casting though attempts were done by designing core. But, due to being highly non-uniform model, the attempt was not successful. So, the cross sections of the bone only made a way to design the outer cortex profile. The sites of taking cross section were the heights where there are some traces of dissimilarities of shapes from the previous section. Next, the cross sections were captured by camera. Some sample cross sections of femur are given in Fig. 5 and the heights were noted down.

Fig. 3 Sample Cross sections

We took 12 different cross sections but they were not enough and besides there were no way to trace the medullar cavity. So, more intensive method was adopted different from the previous way. We sectioned the entire femur with bend saw carefully at defined length shown in Fig. 4. Figure 5 shows some sectioned face.

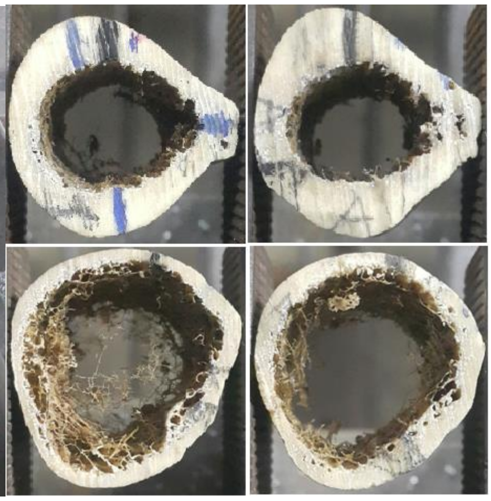


Fig. 4 Sectioning of bone **Fig. 5** Sectioned and named Face

As the bone was brittle so slow cutting operation was precisely followed. From the sections we found the profile of the inner cortex periphery with a bonus of some more extra outer cortex periphery profile. Before starting the main procedure, the sections were named (Fig. 5) and were arranged sequentially so that we can identify and work on each part and of the model separately to obtain a better design.

For capturing the image of the cross sections, a camera was fixed on a vise and the image of each section was captured. The object level was marked on vise and the distance from the camera and the object was always the same. The length was noted down for each section that will be used as the Z co-ordinate value later. The captured images were imported in a software name 'XYit' which can take and generate the co-ordinates of the given points (Fig. 6 and Fig. 7). We tried to take as much points as we could to have the best possible impression of the profile of the outer cortex and the medullar cavity from the cross section image.

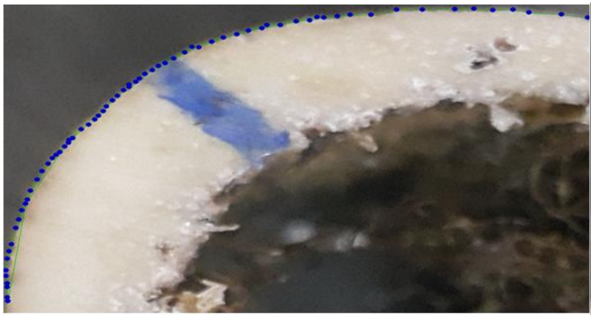


Fig. 6 Point Placing in XYit for outer cortex profile

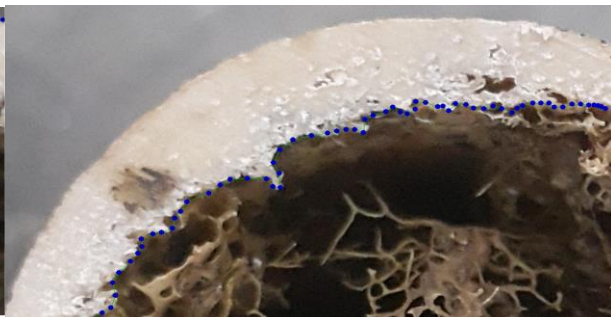


Fig. 7 Point Placing in XYit for inner cortex profile

Later all the data points were extracted as a .dig file where we got the X and Y co-ordinates of the cross sections. As said before, we noted down the Z co-ordinate while capturing the sections. These X Y Z points were pasted in an Excel file where the x and y co-ordinates were multiplied with a zooming factor to convert them to the actual dimension which was initially changed while capturing their photo. These data was then taken in a .txt file that is recognizable by SolidWorks.

The .txt file data were given as input in SolidWorks as 'Curve through XYZ points' and different curves were generated. Fig. 8 shows imported curves of outer and inner cortex. The next figures i.e. Fig. 9 is the image taken after the importation of curves of femur. Guide curves were drawn to loft the curves properly and as close to the real design as possible. Besides for the highly non uniformity of femur and difficulties to take cross-sections, we used some self defined curves for femur.

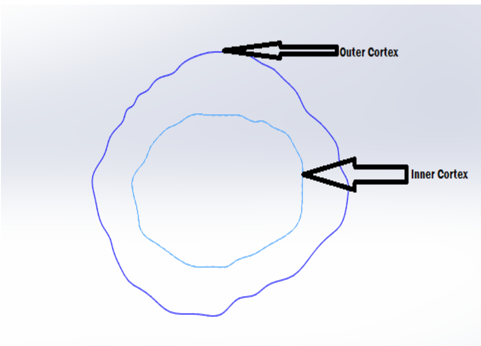


Fig. 8 Importation of the Cross Section Curve

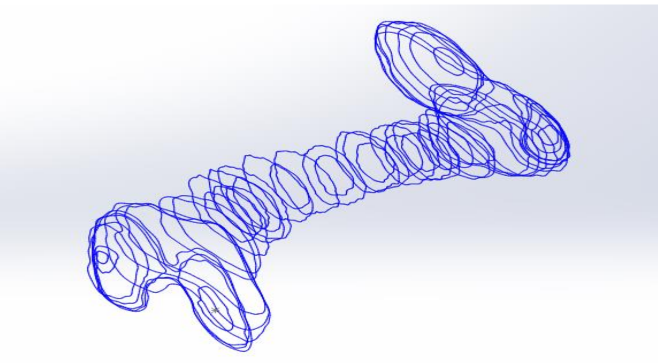


Fig. 9 Curve generation of femur

Then the curves were lofted to get the outer shell of the femur. which has been shown in Fig. 10. The curves of the medullar cavity were used to perform a lofted cut to design the cavity for the femur. It should be mentioned here that the medullar cavity was supposed to be at the part of the bone where the spongy cortex is weak enough to carry load and can be supposed to be void. The cut loft operation is illustrated in Fig. 11. The sectional views in fig 12 shows the completed femur model respectively where medullar cavity is also depicted.

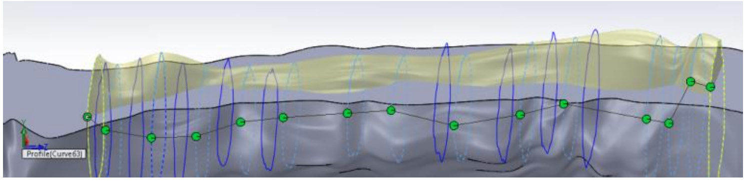


Fig. 11 Cut Loft Opeation

Fig. 10 Femur model

Fig. 12 Sectional view of femur model

The model was then transferred to ANSYS Workbench. The volumetric mesh was generated to the assembled Biomodels taking mesh size 10mm as featured in Fig. 13. The material properties used in this simulation were given in Table 1.

Table 1. Material properties of femur

Young'sModulus(GPa)	Poisson'sRatio	BulkModulus(GPa)	Density(kg/m^3)	ShearModulus(GPa)	Ref.
17.6	0.4	29.3	2000	62.8	[9]

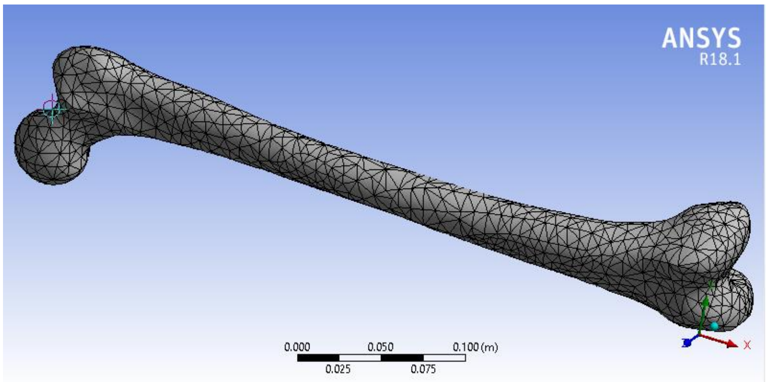
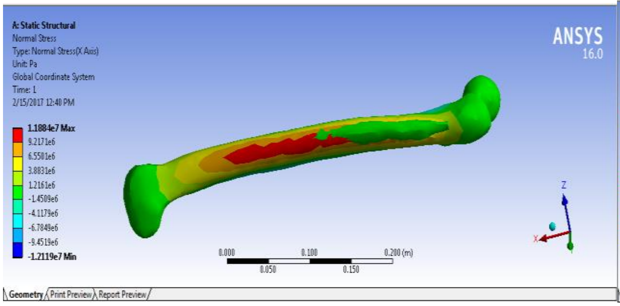


Fig. 13 Meshing of Femur

3. Result and Discussion

Simulation procedure was performed for static and transient conditions. We analyzed the normal stress, shear stress and total deformation. where the exerted force was 300N on each leg assuming the average human weight is 600N . In Fig. 14, normal stress has been depicted and the maximum normal stress of 11.88 MPa has been developed at about 63% of the length of femur from the femur head. In total deformation analysis in Fig. 15 , we observe the maximum total deformation of 2.96mm occurs at the femur head where the neck area exhibit the maximum shear stress of 2.45 MPa (Fig. 16)



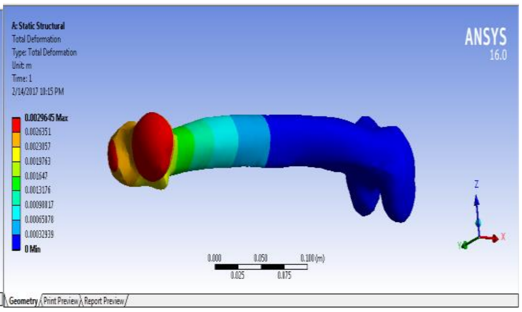


Fig. 14 Normal stress (left side view) along X-axis

Fig.15 Total deformation

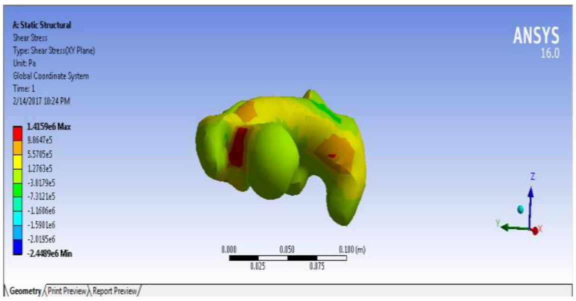
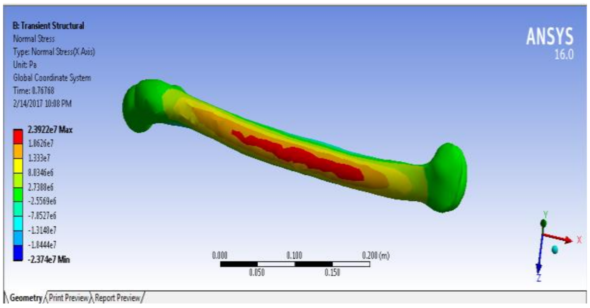


Fig.16 Shear stress (XY) plane (upper neck surface view)

For transient condition, We assume that a complete gait cycle requires 1 second where from 0.1s to 0.4s no load was applied and for .5s to 1s, full load of 300N was applied.



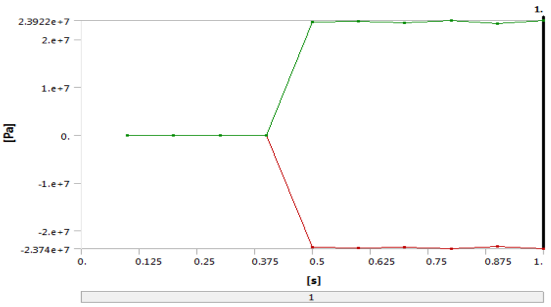
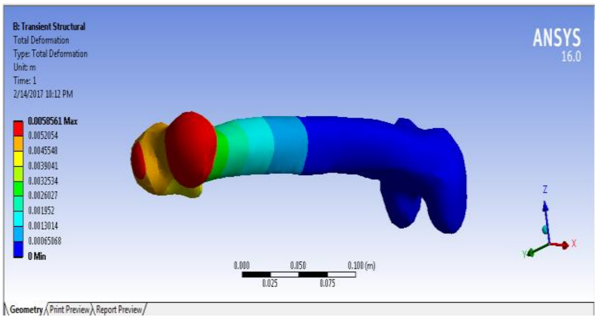


Fig. 17 Normal stress (left side view) along X-axis

Fig.18 Normal stress vs time

In Fig. 17, normal stress has been illustrated where the maximum value is 23.93 MPa and the gamut of the maximum stress zone is between the 25% to 75% length of the femur. Fig. 18 shows the relation of normal stress with time.



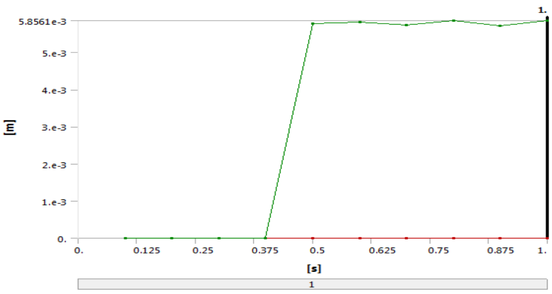
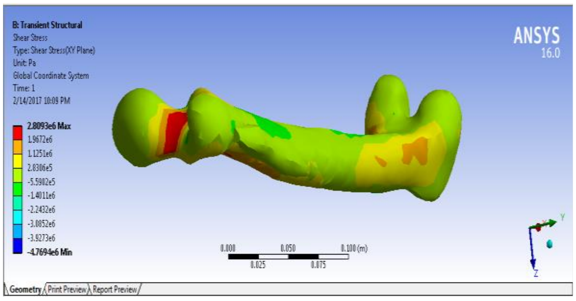


Fig.19 Total deformation

Fig. 20. Total deformation vs time

Fig. 19 marks the maximum total deformation zone that is the head of the femur having the maximum deformation of 5.86 mm. Fig. 20 is the relation of total deformation with time. The maximum shear stress zone that is found from Fig. 21 is the femur neck with the maximum value of 4.7 MPa. Fig .22 shows how the shear stress changes with time.



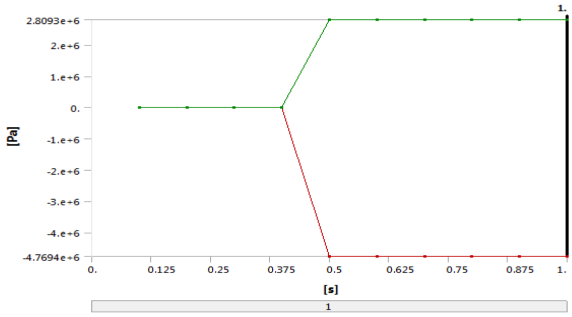


Fig. 21 Shear stress (upper surface view) (XY plane)

Fig. 22 Shear stress vs time

The attempt to create a 3-D solid model of femur has been made successfully. The model was created by making a pattern of Aluminum; the pattern and bone was then cross sectioned and the profile of inner and outer cortex was extracted and then modeled in SolidWorks. A slight deviation in results could be the results of change or difference in the geometric model created or a change in material properties considered for the model. From the simulation, it is observed that the values for the normal stress and total deformation is larger for the transient analysis than the static analysis. The area of the maximum normal stress and total deformation is also larger for transient analysis.

4. Conclusion

The process used to create 3-D model of femur is a novel one and it should be of versatile application that can be adopted to design any type of uneven shaped body. From all above simulated results we can conclude that most vulnerable part of the femur is the femur head, neck and the lower quarter area. Design of femur was done manually. Taking more cross sections would accomplish much more realistic design for the model. For better design of prosthetic leg, the elastic field analysis on femur should be extended to consider the effect of patella, tibia and fibula via a knee joint along with the influence of muscles. Besides, material in the design is expected to be distributed in a heterogeneous fashion.

5. Reference

[1] Huiskes, R. and Chao, E. Y. S., **"A Survey Of Finite Element Analysis In Orthopedic Biomechanics: The First Decade",** Journal of Biomechanics, Vol. 16, 1983, pp. 385-409

[2] 1977-Hayes, W. C. , Swenson L. W. and Schurman D. J., **"Axisymmetric Finite Element Analysis Of The Lateral Tibial Plateau",** Journal of Biomechanics, Vol. 11, 1978, pp. 21-33

[3] A.Özkan, Y. Kişioğlu, **"The rotational loading capacity and effects of Fibula on lower extremity"**,Journal of Engineering Research and Applied Science, Volume 2(1) June 2013, pp. 138-143

[4] Mustafa Ozen*, OnurSayman, Hasan Havitcioglu**"Modeling and stress analyses of a normal foot-ankleand a prosthetic foot-ankle complex"**, Actsof Bioengineering and Biomechanics, Vol. 15, No. 3, 2013.

[5 Popa D, Gherghina G, Tudor M and Tarnita D (2006), **"A 3d GraphicalModelling Method for Human Femur Bone",** *JIDEG*, Vol. 1, pp. 37-40.

[6] Francis A and Kumar V (2012),**"Computational Modeling of Human Femur Using CT Data for Finite Element Analysis",** International Journal of Engineering Research & Technology(IJERT), Vol. 1, No. 6.

[7] McNamara B.P., Cristofolini l., Toni A., Taylor D. (1997)**Journal of Biomechanics,** 30(6), 621-630.

[8] Ghosh, M., Chowdhury, B U, Parvej, M S., A.M. Afsar, *Modelling and Stress Analysis of Tibia-Fibula*, **International Conference on Mechanical Engineering and Applied Science 2017**, February 2017, MIST, Dhaka.

[9] Reilly, D. T., and Burstein, A. H. (1975), The elastic and ultimate properties of compact bone tissue, J. Biomech. 8:393–405.