Finite Element Analysis of Human Femur Bone

U N Mughal[1], H A Khawaja[2*], M Moatamedi[1], M Souli[3]

1. Narvik University College, Norway

2. UiT-The Arctic University of Norway, Norway

3. University of Lille, France

*Corresponding author coordinator@multiphysics.org

Abstract

An effort is made to analyse the stresses experienced by the human femur. In order to achieve these results a CAD model was developed by using the 3-D scanning of generic human femur for an individual of 70kg weight (approx. averaged adult weight). The marrow cavity has been approximated as a hollow cylinder. The FEM model was built using solid tetrahedral element (20-noded 186 structural solid, ANSYS®). The model was analysed for its sensitivity. The results were computed for the range of loads. In this analysis, the maximum stress and its location were noted. In addition, the critical value of load was estimated for ultimate failure (i.e. fracture). The evaluated results give an understanding of the natural safety factor. The presented results are of significant importance in replication of the natural design parameters in creating the synthetic bone substitutes.

1. INTRODUCTION

1.1 Finite Element Method

Finite element method (FEM) is a technique of solution of the boundary value problems. It can be explained as a numerical method for solving differential and integral equations. Finite element analysis (FEA) is the practical application of FEM. FEA is a computational tool for carrying out engineering analysis. It can be used for analysis of new product designs as well as for the existing designs using the equations of mechanics of materials [1, 2]. In this work, FEA is used to analyse the human femur bone.

1.2 Femur Bone

Femur bone is also known as thigh bone. The femur bone is the longest, heaviest and strongest bone in the human body. The length of this bone is almost 26% of the height of person. Femur bone is divided into three parts: upper extremity, body and lower extremity. Upper part consists of head, neck and the tow trochanters. Body is the long and almost cylindrical in shape. It is slightly arched. Lower extremity is bigger than upper extremity. It is slightly cuboid in form but its diagonal diameter is bigger than its anteroposterior. Human skeleton is shown in Figure 1, where femur bones are marked. Bone material has been studied in depth by many workers for e.g. [3-8].

1.3 FEA on Femur bone

FEA is one of the common techniques to examine the structural stresses developed in engineering mechanics [9]. It has been used in many engineering applications including the orthopaedic biomechanics to calculate the stresses in human bones [10, 11]. FEA helps in identifying the zones of high stresses and assist in implants design. Femur bone differs from human to human in both terms of bone geometry and also in the mechanical properties which make it very hard to extract the experimental results to be reproduced. So to analyse the human femur an alternate approach is to use an artificial femur of identical geometry which have approximately same material properties like human bone. This standard geometric model may help to perform experiments to obtain useful results. However, use of modern techniques reduces this complication. Latest techniques of 3D scanning can be employed in building computer aided design (CAD) model of femur bone. The CAD model can be used to build FEM model (mesh of nodes and elements for analysis). FEA analysis can be repeatedly performed on this model with different set of loading conditions and material properties. This methodology is used in given work.

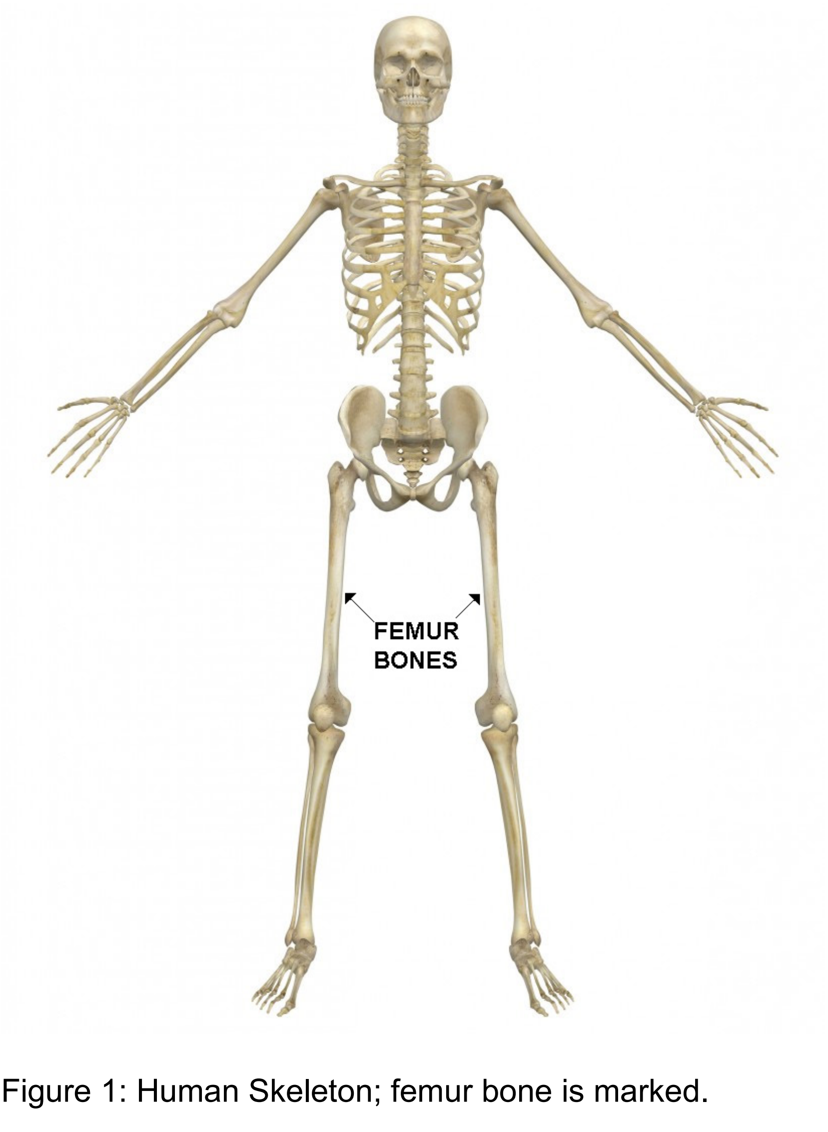

Figure 1: Human Skeleton; femur bone is marked.

It is essential to use the correct material properties and geometric size to simulate the mechanical behaviour of wide range of bone quality and size. The aim of this study is to give the structure for bone stiffness and strength suitable for use within the framework.

In this study, two different kinds of loading conditions are applied on the femur bone model; in axial direction (parallel to bone) and in bending direction (normal to the bone). Appropriate assumptions are taken for this study. The main theme of this study is to create a simulation model that can demonstrate the stresses and strains which may happen on real bone. The accuracy of model is verified and compared with results available in literature [9, 12 and 13].

2. MATERIAL AND METHOD

2.1 Methodology

Two different specimens of human femur bones were collected for this study. Given specimens were from the subjects of around 70 Kg in weight.

2.2 3-D Laser Scan

3-D laser scanning was performed on available specimens using NextGen® 3D laser scanner. From this procedure data cloud was obtained which can be imported in CAD software to obtain the geometric features of a femur bone. Obtained data cloud is shown in Figure 2.

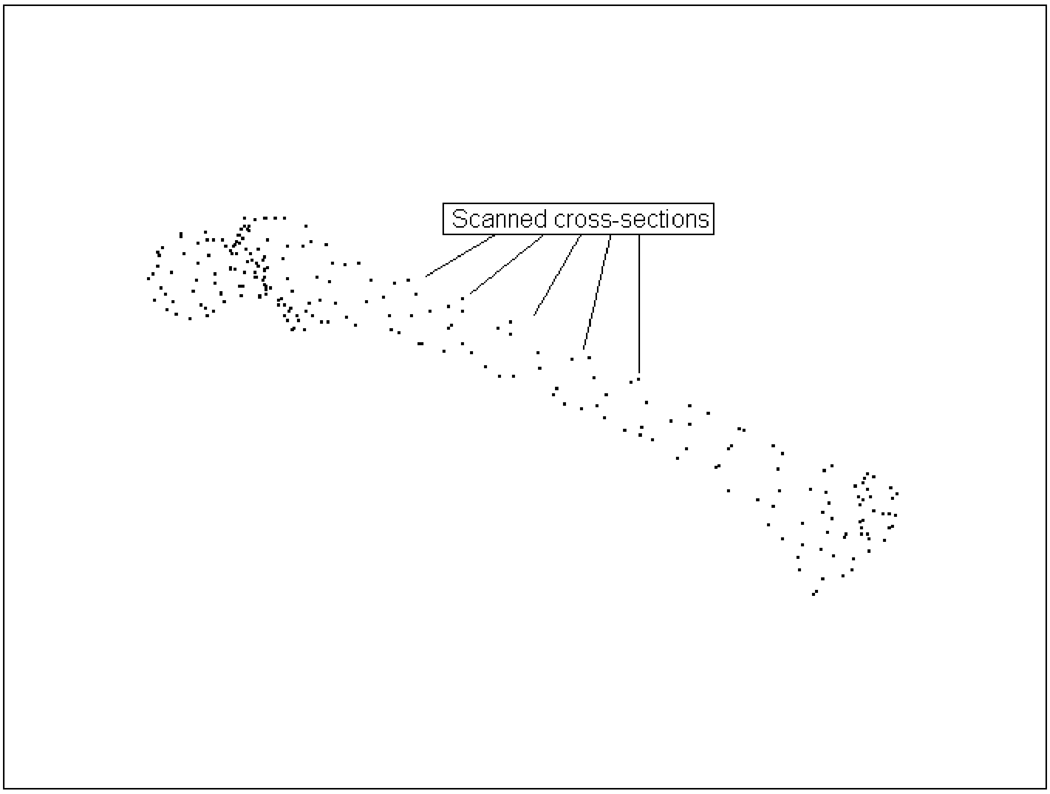

Figure 2: Data Cloud of Human Femur from 3-D Scanner

2.3 CAD Model

Femur CAD model was developed in SolidWorks® using the technique of transforming 2D geometry into 3D using the cloud data obtained earlier [14, 15]. The CAD model of the Femur bone is shown in Figure 4. At this stage CAD model was obtained based on external geometric features of femur bone however, the internal detail (e.g. marrow cavity) in CAD was approximated.

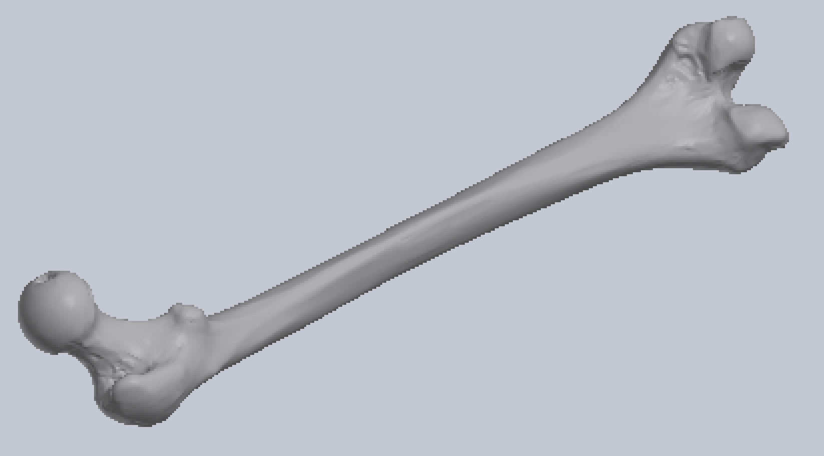

Figure 3: CAD Model of Human Femur (SolidWorks®)

The dimensions of marrow cavity are hard to know considering it is hallow imprint within the bone. The dimensions of marrow cavity were approximated in this work with a hallow cylinder following the curvature of the bone with spherical ends. The diameter of marrow cavity was taken as 1.6 cm. Approximated marrow cavity is shown in Figure 4.

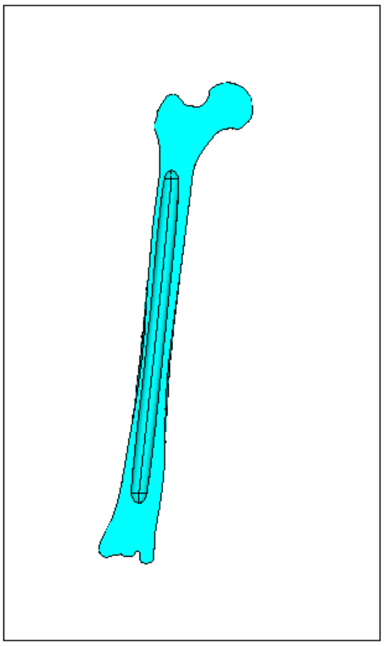

Figure 4: Marrow cavity is hallowing cut curving cylinder of radius 1.6 cm with spherical ends.

2.4 FEA Mesh

FEA mesh represents the nodes and elements for structural calculations. In this work, FEA mesh was generated on 3D CAD model of femur bone. Tetrahedral 20-noded 186 Structural Solid elements were used to build the FEA mesh [13]. The stated element type was chosen since it has higher accuracy compared to its equivalent lesser node element [13]. FEA mesh was generated using the auto-mesh generation algorithm in FEA software ANSYS® [9, 12, 13]. Mesh refinement was performed in desired segments of the bone to avoid unrealistic stress concentration points. Furthermore mesh was refined in regions of higher gradients to magnify the accuracy of results. Mesh sensitivity analysis was also carried out to ensure the quality of results. FEA mesh on femur bone is shown in Figure 5.

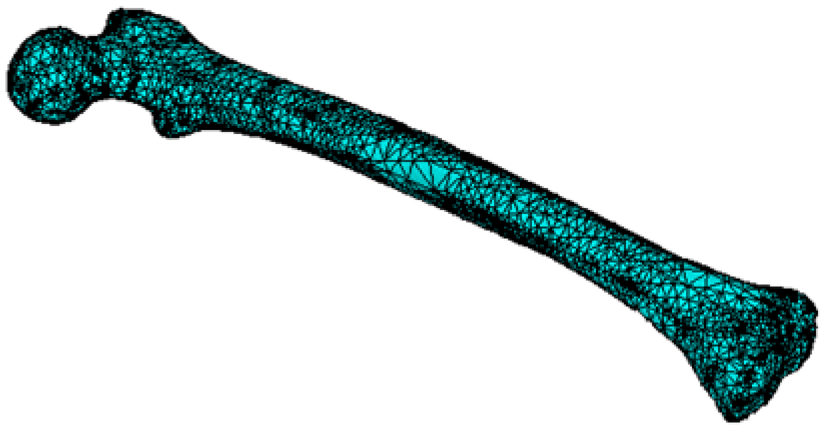

Figure 5: FEA Mesh of Femur Bone

2.5 Material Properties

Material properties of human femur vary between subjects therefore and it is difficult to assign any particular material properties. Furthermore, the bone material is anisotropic in nature, however it was assumed to be isotropic because complete femur bone was taken for analysis. The justification is that a small segment of bone can be solved for anisotropic solution however for complete bone; it is extremely difficult to assign anisotropic material properties.

The young's modulus of the femur bone varies from 10 to 20 GPa and for the analysis it was taken to be 15 GPa. Poisson's ratio and density used were 0.3 and 2000 Kg/m^3 respectively [3]. Linear elastic material model was chosen for this analysis.

Linear static analysis was performed and physiological conditions (role of muscles in sharing the load) were ignored.

2.6 Loading and Boundary conditions

Two different kind of loading conditions were applied to simulate real case scenarios. In first case axial loading (compression) was applied in direction of the bone. This case simulates the weight handled by femur in upright standing position. In second case bending load (perpendicular) is applied to femur bone. In both cases boundary constraint was applied on the other end of the femur bone. Loading and boundary constraints are shown in Figure 6:

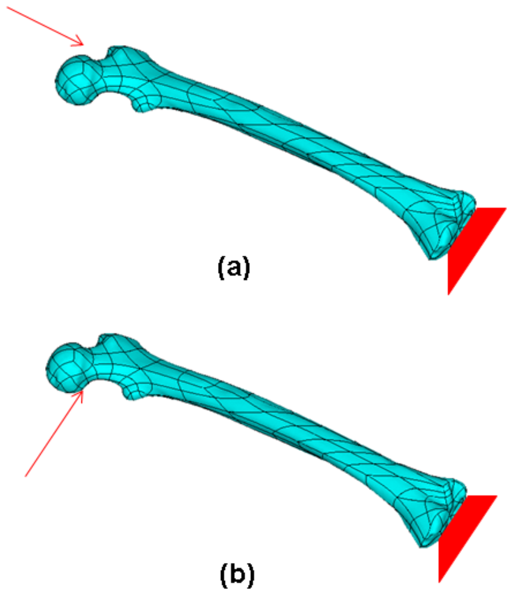

Figure 6: (a) Axial loading condition with boundary constraint on other end. (b) Bending loading condition with boundary constraint on the other end

3. RESULTS

FEA analysis is performed on model of femur bone by varying the loads. The maximum stresses generated in this analysis are given in Table 1. The failure stress is taken as 100 MPa based on experimental data available in literature [4].

Table 1: Summary of Stresses from Axial and Bending Loadings; * indicates failure

Loading (Kg)	**Max. Stress (Axial)-MPa**	**Max. Stress (Bending)-MPa**
25	6.04	36.31
35	9.31	51.22
69	24.23	100.30*
150	36.41	217.93
250	60.01	363.11
414	99.83*	601.93
500	121.12	733.31

The results indicate the failure of femur bone under the loading of 414 Kg of weight under axial loading and 69 Kg of load under the bending load. The results clearly indicate that the strength of femur bone in axial direction is significantly more than compared to bending. The point of high stress is indicated in Figure 7.

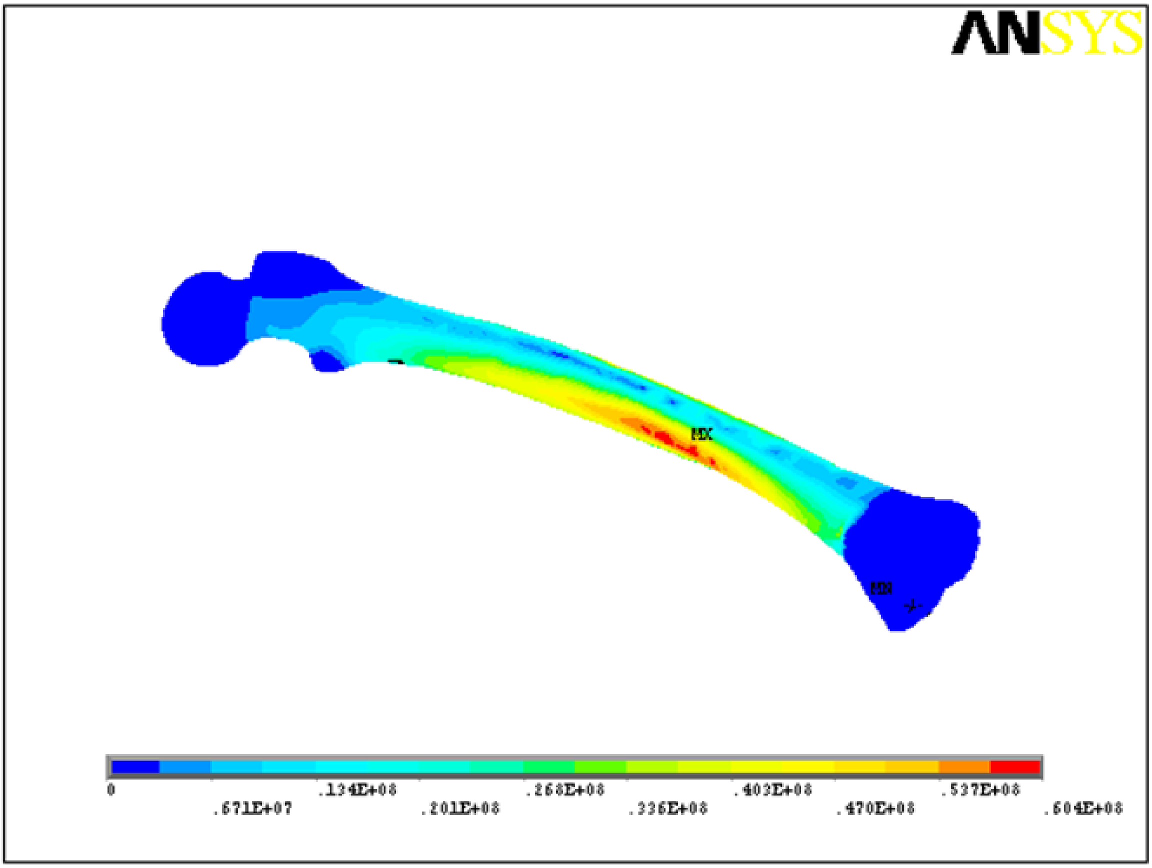

Figure 7: High Stress location in Femur Bone under Axial Loading

4. CONCLUSIONS

Following conclusions can be drawn from FEM analysis of human femur bone:

1 Axial strength of femur is almost six times than bending.

2 Human femur can withstand ten times the load of its body weight.

3 Evaluated results are the indicative of the failure criteria of substitute material for bones.

4 Given methodology can be used over other biomechanical structures for study.

REFERENCES

[1] Szabo, B.A. and I. Babuška, Finite Element Analysis 1991: Wiley.

[2] Zienkiewicz, O.C., R.L. Taylor, and J.Z. Zhu, The Finite Element Method: Its Basis and Fundamentals: Its Basis and Fundamentals 2005: Elsevier Science.

[3] Burstein, A.H., et al., The ultimate properties of bone tissue: The effects of yielding. Journal of Biomechanics, 1972. 5(1): p. 35-44. DOI: http://dx.doi.org/10.1016/0021-9290(72)90017-6

[4] Ji, B. and H. Gao, Mechanical properties of nanostructure of biological materials. Journal of the Mechanics and Physics of Solids, 2004. 52(9): p. 1963-1990. DOI: http://dx.doi.org/10.1016/j.jmps.2004.03.006

[5] Currey, J.D., Mechanical properties of bone tissues with greatly differing functions. Journal of Biomechanics, 1979. 12(4): p. 313-319. DOI: http://dx.doi.org/10.1016/0021-9290(79)90073-3

[6] Currey, J.D., The effect of porosity and mineral content on the Young's modulus of elasticity of compact bone. Journal of Biomechanics, 1988. 21(2): p. 131-139. DOI: http://dx.doi.org/10.1016/0021-9290(88)90006-1

[7] Katz, J.L. and Y. Hyo Sub, The Structure and Anisotropic Mechanical Properties of Bone. Biomedical Engineering, IEEE Transactions on, 1984. BME-31(12): p. 878-884. DOI: http://dx.doi.org/10.1109/TBME.1984.325252

[8] Weiner, S. and H.D. Wagner, THE MATERIAL BONE: Structure-Mechanical Function Relations. Annual Review of Materials Science, 1998. 28(1): p. 271-298. DOI: http://dx.doi.org/10.1146/annurev.matsci.28.1.271

[9] Stolarski, T., Y. Nakasone, and S. Yoshimoto, Engineering Analysis with ANSYS Software 2011: Elsevier Science.

[10] Papini, M., et al., The biomechanics of human femurs in axial and torsional loading: comparison of finite element analysis, human cadaveric femurs, and synthetic femurs. J Biomech Eng, 2007. 129(1): p. 12-9. . DOI: http://dx.doi.org/10.1115/1.2401178

[11] Huang, B.W., et al., Dynamic Characteristics of a Hollow Femur. Life Science Journal, 2012. 9(1): p. 723-726.

[12] ANSYS®, Academic Research, release 12.0.

[13] ANSYS®, Academic Research, Theory Reference, in Structures, Static Analysis release 12.0.

[14] Systèmes, D., SolidWorks®, release 2012.

[15] Lombard, M., Solidworks 2013 Bible 2013: Wiley.