

A STUDY ON BIOMECHANICAL STRUCTURE-FEMUR BONE

Prof.Karthik. A. S¹, Akshay. A. K², Keertiraj. M. R³, Abhineet. K⁴, Kudalappa. G⁵

¹Prof.Karthik A.S, Professor, Mechanical department, S.K.S.V.M.A.C.E.T, Karnataka, India

²Akshay. A. K, Student, Mechanical department, S.K.S.V.M.A.C.E.T, Karnataka, India

³Keertiraj.M. R, Student, Mechanical department, S.K.S.V.M.A.C.E.T, Karnataka, India

⁴Abhineet. K, Student, Mechanical department, S.K.S.V.M.A.C.E.T, Karnataka, India

⁵Kudalappa. G, Student, Mechanical department, S.K.S.V.M.A.C.E.T, Karnataka, India

ABSTRACT

An effort is made to analyze 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 FEM model was built using solid tetrahedral element (20-noded 186 structural solid, ANSYS). The model was analyzed 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.

Keyword: - Biomechanical, Femur Bone.

1. INTRODUCTION

1.1 Finite element method

The Finite component strategy (FEM) is a procedure of arrangement of the limit esteem issues. It can be clarified as a numerical strategy for settling differential and basic conditions. Limited component investigation (FEA) is the down to earth use of FEM. FEA is a computational instrument for completing building investigation. It can be utilized for examination of new item plans and in addition for the current outlines utilizing the conditions of mechanics of materials. In this work, FEA is utilized to investigate the human femur bone.

1.2 Femur Bone

Femur bone is otherwise called thigh bone. The femur bone is the longest, heaviest and most grounded bone in the human body. The length of this bone is right around 26% of the stature of individual. Femur bone is isolated into three sections: furthest point, body and lower limit. Upper part comprises of head, neck and the tow trochanters. Body is the long and relatively tube shaped fit as a fiddle. It is marginally curved. Lower furthest point is greater than furthest point. It is marginally cuboids in shape yet its askew distance across are greater than 1, where femur bones are stamped. Bone material has been examined top to bottom by numerous specialists for FEA ON FEMUR BONE .FEA is one of the regular methods to look at the auxiliary burdens created in designing mechanics. It has been utilized as a part of many designing applications including the orthopedic biomechanics to ascertain the worries in human bones. FEA helps in distinguishing the zones of high burdens and aid inserts outline [1].

Femur bone contrasts from human to human in the two terms of bone geometry and furthermore in the mechanical properties which make it difficult to separate the exploratory outcomes to be replicated. So to break down the human femur a substitute approach is to utilize a manufactured femur of indistinguishable geometry which has roughly

same material properties like human bone. This standard geometric model may perform investigations to acquire helpful outcomes. Be that as it may, utilization of current procedures decreases this entanglement. Most recent methods of 3D checking can be utilized in building PC helped plan (CAD) model of femur bone. The CAD model can be utilized to assemble FEM show (work of hubs and components for examination). FEA examination can be over and again performed on this model with various arrangement of stacking conditions and material properties. This technique is utilized as a part of given work.

It is basic to utilize the right material properties and geometric size to reproduce the mechanical conduct of extensive variety of bone quality and size. The point of this examination is to give the structure for bone solidness and quality appropriate for use inside the system. In this investigation, two various types of stacking conditions are connected on the femur bone model; in hub bearing (parallel to bone) and in twisting course (ordinary deep down). Suitable suspicions are taken for this investigation. The fundamental topic of this examination is to make a reenactment show that can exhibit the anxieties and strains which may occur on genuine bone. The precision of model is confirmed and contrasted and comes about accessible in writing.

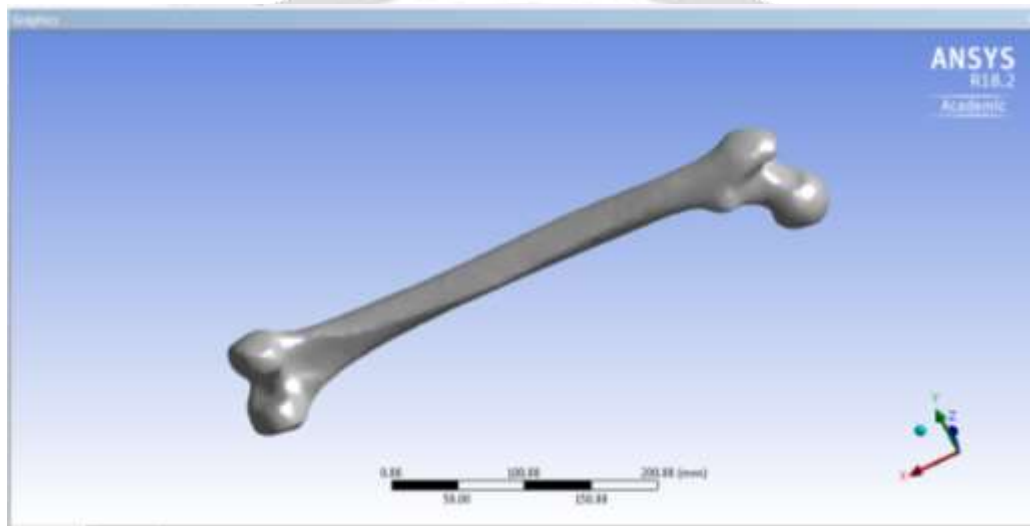


Fig -1: 3D extracted model of femur bone.

1.3 Material and Method

1) Strategy

Two distinct examples of human femur bones were gathered for this examination. Given examples were from the subjects of around 70 Kg in weight.

2) Computer Aided Design Model

Femur CAD demonstrates was created in Solid Works utilizing the strategy of changing 2D geometry into 3D utilizing the cloud information got before. The CAD model of the Femur bone is appeared in Figure 3. At this stage CAD demonstrates was gotten in view of outer geometric highlights of femur bone be that as it may, the inward detail (e.g. marrow depression) in CAD was approximated.

The measurements of marrow hole are difficult to know thinking of it as is honor engrave inside the bone. The measurements of marrow cavity were approximated in this work with a praise chamber of span 1.6 cm with round finishes barrel following the ebb and flow of the bone with circular closures.

2. 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. The stated element type was chosen since it has higher accuracy compared to its equivalent lesser node element. FEA mesh was generated using the auto-mesh generation algorithm in FEA software ANSYS®. 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 of femur bone is shown in Figure.

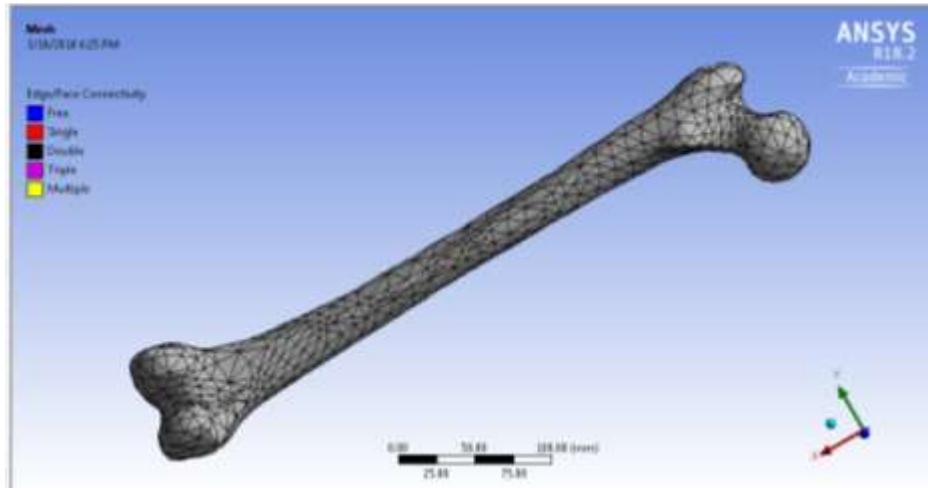


Fig -2: Tetrad meshing of femur bone

3. 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 20GPa and for the analysis it was taken to be 15GPa. Poisson's ratio and density used were 0.3 and 2000 Kg/m³ respectively. Linear elastic material model was chosen for this analysis. [1]

4. LOADING AND BOUNDARY CONDITIONS

The individual stature weight is handled by the femur bone in upright position considering this as the main object we apply the boundary conditions, using the Ansys software we fix the one end of the femur bone i.e. which is in contact with the knee bone, is fixed with faces defined by the report and the other end i.e. which is in contact with the pelvic girdle, with certain confined faces and these faces are subjected to the moment gradually increasing the moment load and by the analysis we get breakdown point, this load is considered as the maximum sustainable torisonal load by the femur bone.

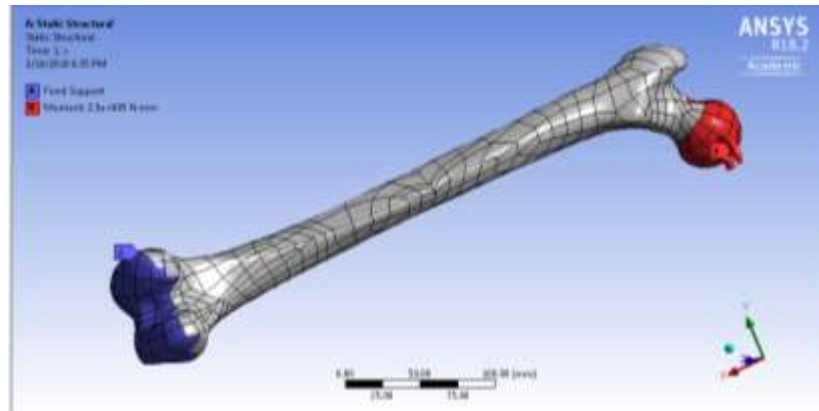


Fig -3 Boundary conditions

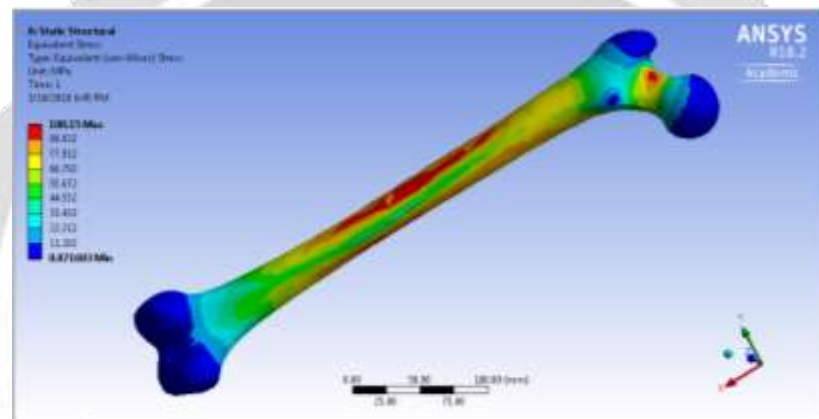


Fig -4: Maximum torsion stress

LOADING (Nm)	MAX.STRESS(MPA) TORISIONAL
100	42.08
200	84.161
230	96.785
235	98.889
<u>237.8</u>	<u>100.07</u>
240	100.99
250	105.2

Table -1: Torsion Result Analysis of femur bone.

5. RESULT

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 100MPa based on experimental data available in literature [4].

The results indicate the failure of femur bone under the loading of 240Nm of load under torsion load. The results clearly indicate that the strength of femur bone in torsional direction is significantly less when compared to other analysis such as bending and axial. The point of high stress is indicated in Figure 4.

6. CONCLUSION

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 100MPa based on experimental data available in literature.

- The results indicate the failure of femur bone under the loading of 240Nm of load under torsion loading.
- The results clearly indicate that the strength of femur bone in moment direction is significantly less than compared to bending and axial loading.
- The point of high stress is indicated in Figure 3 and 4.
- Human femur can withstand ten times the load of its body weight.
- Evaluated results are the indicative of the failure criteria of substitute material for bones.
- Given methodology can be used over other biomechanical structures for study.

7. ACKNOWLEDGEMENT

The Present work has been successfully completed by the support and guidance of our chief admin Dr.Srinivas.Rajput and I want to extend my gratitude to beloved principal Dr.Hamppanavar for his assistance in carrying out the analysis.

8. REFERENCES

- [1] Uzair N. Mughal, Hassan A. Khawaja and M. Moatamedi, Finite element analysis of human femur bone, Int. Jnl. of Multiphysics Volume 9 · Number 2 · 2015
- [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.
- [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.
- [5] Currey, J.D., Mechanical properties of bone tissues with greatly differing functions. *Journal of Biomechanics*, 1979. 12(4): p. 313–319.
- [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.
- [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)
- [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.
- [9] Stolarski, T., Y. Nakasone, and S. Yoshimoto, *Engineering Analysis with ANSYS Software 2011: Elsevier Science. Int. Jnl. of Multiphysics Volume 9 · Number 2 · 2015* 107
- [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.
- [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*.