A Study on Biomechanical Structure (Femur Bone)

Prof.Karthik A.S.[1] Akshay A.[2] Keertiraj M. R.[3] Abhineet. K.[4] Kudlappa G.[5]
[1,2,3,4,5]Smt Kamala & Sri Venkappa M. Agadi College of Engineering & Technology, Laxmeshwar, India

Abstract— An exertion is made to break down the burdens experienced by the human femur. Keeping in mind the end goal to accomplish these outcomes a CAD show was created by utilizing the 3-D checking of bland human femur for a person of 60 kg weight. The marrow pit has been approximated as an empty chamber. The FEM demonstrate was constructed utilizing strong tetrahedral component (20 noded, ANSYS workbench 14.5). The model was broke down for its affectability. The outcomes were processed for the scope of burdens. In this examination, the most extreme pressure and its area were noted. Furthermore, the basic estimation of load was evaluated for extreme disappointment (i.e. break). The assessed comes about give a comprehension of the characteristic wellbeing factor. The displayed comes about are of noteworthy significance in replication of the regular outline parameters in making the manufactured bone substitutes.

Key words: Biomechanical, Femur Bone

I. Introduction

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.

A. 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 is greater than1, 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 have 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.

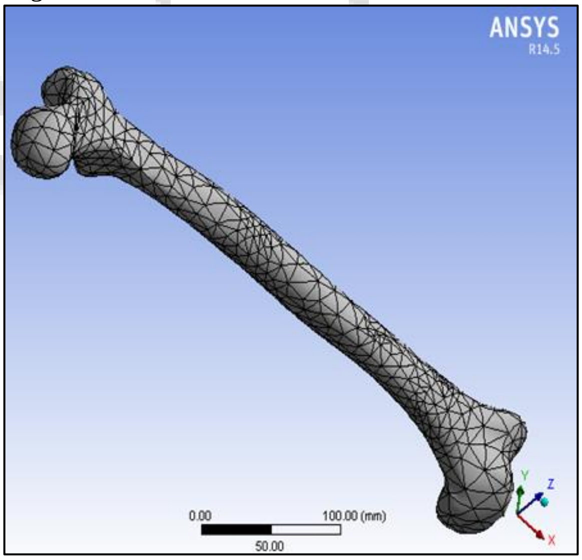

Fig. 1: Human Skeleton; Femur Bone Is Marked

B. 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 demonstrate 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 demonstrate 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. Across of

II. 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.

III. 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. [1]

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/m3 respectively. Linear elastic material model was chosen for this analysis. [1]

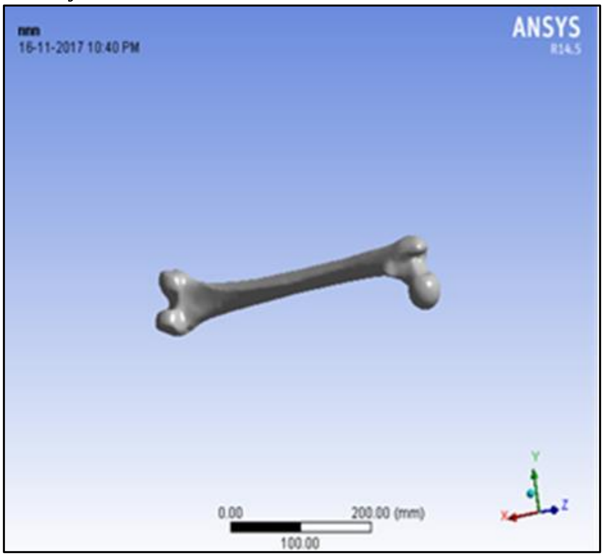

Fig. 2: Model Generated in CATIA

IV. 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

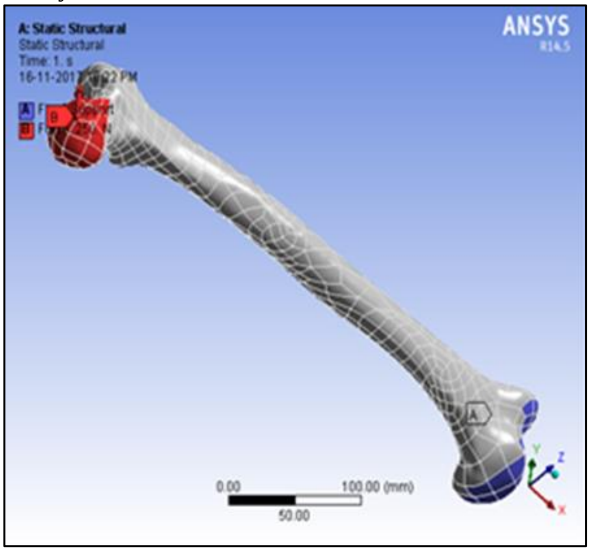

Fig. 3: Boundary Conditions

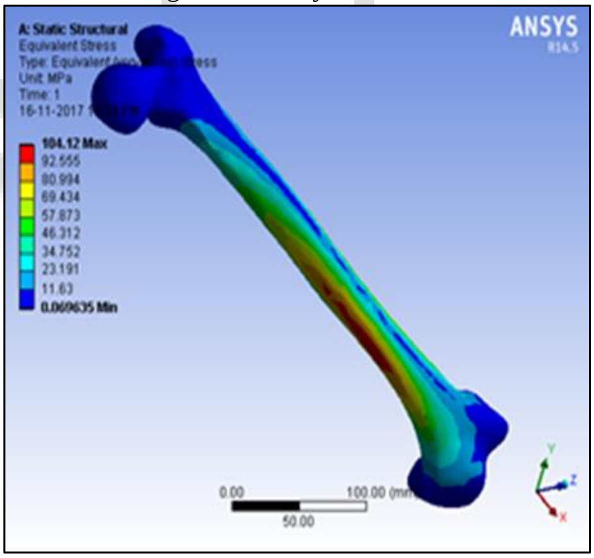

Fig. 4: Max. Axial Stress

Loading (N)	Stress Axial(MPa)	Stress Bending(MPa)
250	5.344	28.921
350	7.0488	40.489
700	14.09	80.979
900	19.247	104.12
1500	32.069	173.53
2500	53.448	289.21
4150	88.724	480.09
5000	106.9	578.42

Table 1.Result Analysis

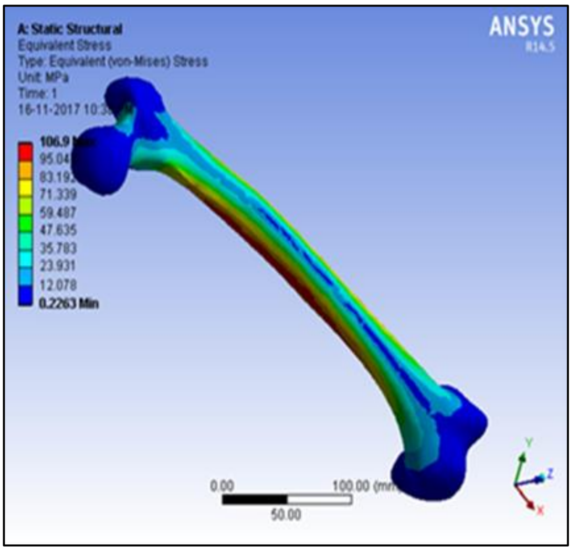

Fig. 5: Max Bending Stress

V. 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 100 MPa based on experimental data available in literature.

- The results indicate the failure of femur bone under the loading of 500KG of weight under axial loading and 90 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 4 and 5.
- Axial strength of femur is almost six times than bending.
- 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.

VI. ACKNOWLEDGMENT

The Present work has been successfully completed by the support and guidance of our head Dr.Devendra K. and I want to extend my gratitude to beloved principal Dr.S.V.Gorabal for his assistance in carrying out the analysis.

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): p. 878–884.

[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.