International Journal of Science and Research (IJSR)
ISSN (Online): 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2017): 7.296

Modelling and Static Analysis of Femur Bone by Using CAE

Nagaraja S N[1], N Arun Kumar[2], P G Sunil Kumar[3]

[1]Asst. Professor, Dept. of Mechanical Engineering, Kuppam Engineering College, Andhra Pradesh

[2]Student, Dept. of Mechanical Engineering, Kuppam Engineering College, Kuppam

[3]Student, Dept. of Mechanical Engineering, Kuppam Engineering College, Kuppam

Abstract: ***Femur is leg bone of the human body undergoing more deformation. Biomechanics is the theory of how tissues, cells, muscles, bones, organs and the motion of them and how their form and function are regulated by basic mechanical properties. The aim of this study is to create a model of real proximal human femur bone and the behaviour of femur bone is analyzed in ANSYS under physiological load conditions. A finite element model of bones is generated by using CT scan data are being widely used to make realistic investigations on the mechanical behaviour of bone structures. Orthopedic implantation is done in case of failure. Before implantation it is necessary to analyze the perfectness in case of its material property, size and shape, surface treatment, load resistance and chances of failure. Analysis is done for the stresses formed in different femur implant materials under static loading condition using ANSYS software. Analysis is done on different materials like structural steel, Aluminium Oxide and Ti-6Al-4V implant materials. Since each femur carries 1/2 the body weight, analysis is done for 245 N load, including the cases of patient carrying certain weight. And based on the analysis it can be concluded that, while comparing these two implant materials Ti-6Al-4V gave less deformation on static load conditions. TI-6AL4V is a low density material, which has excellent bio compatible and mechanical properties, it is ideal for the use of an implant in surgeries. Finally the success of implantation depends on implant material and size, implantation method and its handling by the patient.***

Keywords: Biomechanics, CATIA, FEA, Femur

1. Introduction

Biomechanics is the application of mechanical principles on living organisms. By applying the laws and concepts of physics, biomechanical mechanisms and structures can be simulated and studied. Finite Element Method (FEM) is widely accepted as a power tool for biomechanics modeling. Irregular geometry, complex microstructure of biological tissues and loading situations are specific problems of the FEM in biomechanics and are still difficult to model. Straight beam theory is proposed to calculate stress distributions in the femur due to the body weight and some muscles force given some major simplifying assumptions on the muscles and the joint reactions. FE model would be advantageous in complementing experimental works and in overcoming the inherent limitations associated with experimental studies which can provide only limited amount of information. Although some of these methods were found to provide enough automation, intrinsic accuracy, robustness and generality to be used in clinical applications. Hard tissues are rigid organs that form part of the endoskeleton of vertebrates. Bone tissue is a type of dense connective hard tissue. Bones is composed of inorganic salts impregnated in a matrix of collagen fibers, proteins and minerals. They maintain the shape of body and to assist in force transmission during movement. Long bones are characterized by a shaft, the diaphysis that is much longer than it is wide. The femur bone is the most proximal bone of the leg in vertebrates capable of walking or jumping. In human anatomy, the femur is the longest and largest bone but strongest under compression only. The femur at its bottom portion meshes with the tibia bone to create the knee joint. At its top end, the femur meshes with the acetabulum to create the hip joint. The femur is responsible for bearing the largest percentage of body weight during normal weight-bearing activities. The aim of this study is to create a model of human femur bone in CATIA software. This model was analyzed in FEM package ANSYS. This paper aims to construct a complete three-dimensional Femur bone from CT scan data. The CATIA software is used to create 3D models and smooth the surface of the domain. The Finite element method is applied to find the stress distribution and deformation on different implant materials at different load conditions.

2. Literature Survey

Ajay Dhanopia, Prof. (Dr.) Manish Bhargava, "Finite Element Analysis of Human Fractured Femur Bone Implantation with PMMA Thermoplastic Prosthetic Plate" [1] Femur bone is the longest and stronger bone in the human body. This bone is contain linear elastic, isotropic and homogeneous material of calcium phosphate. It needs to support maximum weight of the body in between hip joint and knee joint during static loading conditions. Bone fracture is one of the common traumas. One method of rectifying the fractured bone is by joining the fractured bone by using prosthetic bone plates and screws. The objective of this study is to finite element analysis (FEA) of human fractured bone fixation with poly-methyl methacrylate thermoplastic (PMMA) prosthetic plate at mid-shaft position in static loading condition. Result analysis based on the thermoplastic biomaterial of which calculated mechanical strength is matched with the nearest value of femur material strengths. To prove that PMMA is best suitable material, compared the minimum value of an equivalent stress (von-misses stress),

ISSN (Online): 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2017): 7.296

maximum total deformation, maximum and minimum principal stress with respect of other biomaterials.

Ashwani Kumar, Himanshu jaiswal, Tarun Garg, Pravin P Patil, "Free Vibration Modes Analysis of Femur Bone Fracture Using Varying Boundary" [2] The main objective of the femur bone analysis is to know the natural frequencies, natural vibration +96modes and identify the fracture location of the bone through the computer simulation based on the FEA. Finite element Method or Finite Element Analysis is an approximation techniques used for the analysis of complex objects and geometries. The femur bone analysis is subjected to free-free and fixed-fixed boundary conditions. For these two different boundary conditions natural frequencies and natural vibration modes are identified. The mode shape shows that the natural frequency of free-free boundary condition varies from 0 Hz to 1381.1 Hz and for fixed-fixed boundary condition 1211 Hz to 7856.4 Hz. On the bases of these two boundary conditions mode shape is determined and fracture location can be easily notified. To prevent the fracture of femur bone external excitation frequency must be avoided to coincide with these natural frequencies. The results were compared with experimental results available in literature. For the design of femur bone model Solid Edge software is used and the model is imported in ANSYS R 14.5 (FEA based software) for the free vibration analysis.

3. Methodology

3.1 Creating the model of the femur bone

For FE analysis of femur bone, firstly the three dimensional model of femur was developed. In the present study we have used an ideal femur bone model as presented in literature. The model used represented an ideal human femur bone with age of 35 years old healthy individual whose weight is 50 Kg which was reconstructed from CT (DICOM) images. Then it was imported to (ANSYS Workbench).

Figure: Femur Posterior View

Then the model is divided into two parts, the upper part which includes the head and neck and the lower part which includes the body of the femur and the condyles, the process of creating two parts is to consider the upper part as cortical bone and the lower part as trabecular bone.

Figure: Catia Model of Femur Bone

3.2 Mesh preparation

Mesh is a very important step required for Finite Element Analysis of the femur model, an optimized mesh has been developed using model wizard in ANSYS Workbench, a proper setting and values have been executed in order to use smaller elements on proximities and curvatures for the model. The numbers of tetrahedral elements used for the femur model are 7888, while the number of nodes is 14186 as shown in figure below.

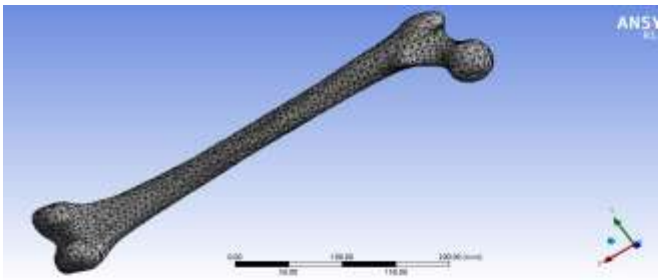


Figure: Mesh Generation

3.4 Assignment of material properties for the model

One of the important parameters necessary for the stress analysis is the assignment of the material properties for the human femur model, we have assigned the material properties according to the values found in literature, the following section presents the major parameters assigned for our model are Density and Modulus of Elasticity.

Table: Material Properties of Bone

Bone Materials	Density (Kg/m^3)	Young's modulus (GPa)	Poisson's Ratio	Ultimate Tensile Strength (MPa)
Alumina Al_2O_3	8500	240	0.31	210-290
Ti-6Al-4V	4500	120	0.32	993

3.3 Boundary conditions

An important step required for the FEA is the boundary condition for the femur bone model, a special procedure has been used to represent the boundary conditions for the femur model during the normal daily activities, A fixed boundary condition has been applied on the distal end of the femur and the hip contact force has been applied on the head of the femur in order to calculate the normal stresses for normal walking and standing up during this boundary condition.

Figure: Fixed Support

Figure: Force

3.5 Loading Conditions

The type of stress analysis used in the present study is the transient structural analysis (also called time history analysis). This type of analysis is used to determine the dynamic response of a structure under the action of any general time-dependent loads. In Transient analysis, the load can be simulated with time dependent values, therefore in the present study we need to measure the hip contact force that applied on the head of the femur during a complete gait cycle for the activities included in my present study. Since 1980's, Bergmann and his research group have been pursuing the instrumented hip implants with telemetric data transmission. These gait data which includes the hip contact force with the cycle duration have been used in my present study for a typical patient weight (50 kg) in order to calculate the stresses on the human femur bone.

4. Results and Discussion

Applied force of 25 kg (245 N) will act in downward x-direction in current case because we assumed that the average weight of adult person is nearly comes of define weight at the femur head in downward direction applied by selecting the circular /circumferential area which shows in below figure's by red color.

4.1 Aluminium Oxide (Al_2O_3)

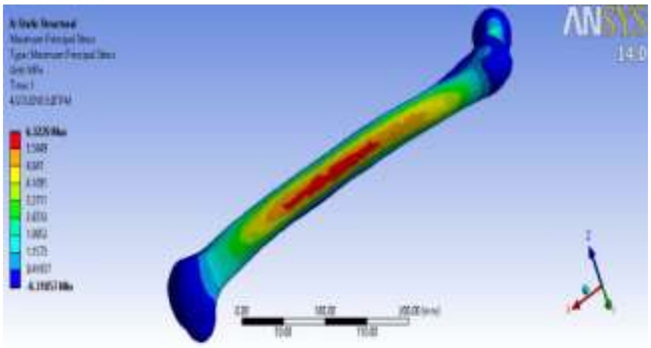


Figure: Shows the distribution of stress during standing

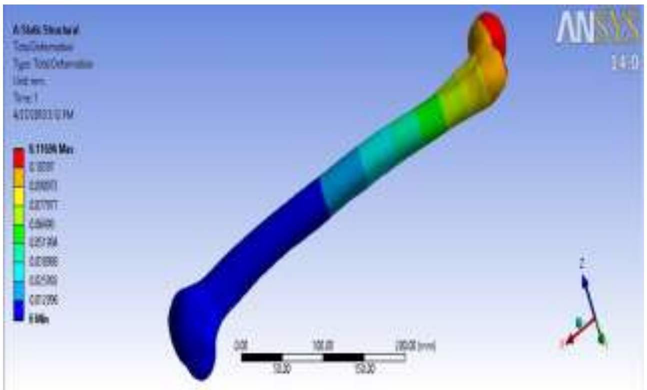


Figure: Shows the distribution of total deformation during standing

Graph: Stress Vs Deformation

The above graph shows the details of stress and deformation of the femur bone during standing

4.2 Titanium Alloy (Ti-6Al-4V)

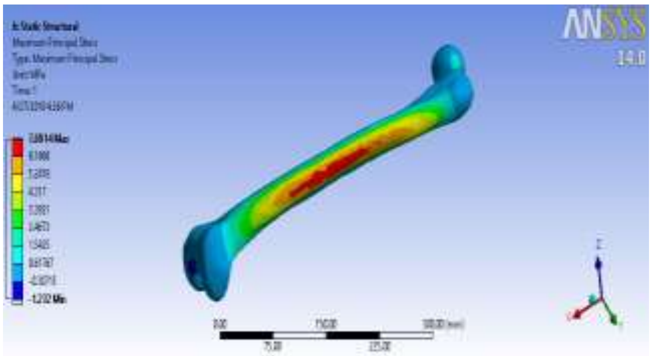


Figure: Shows the distribution of stress during standing

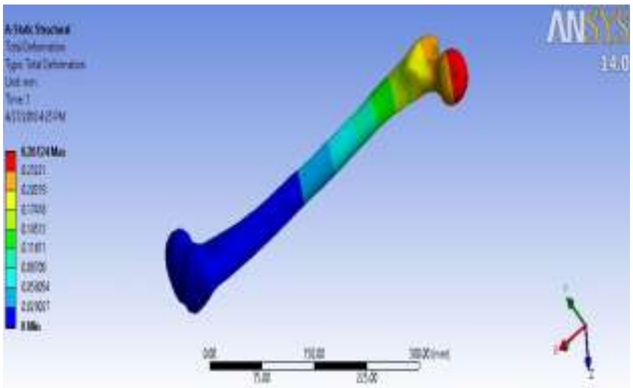
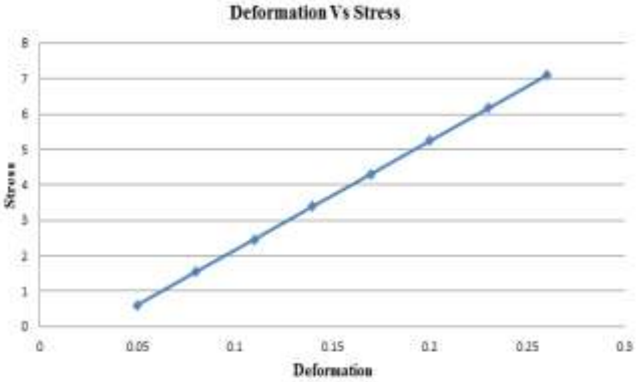
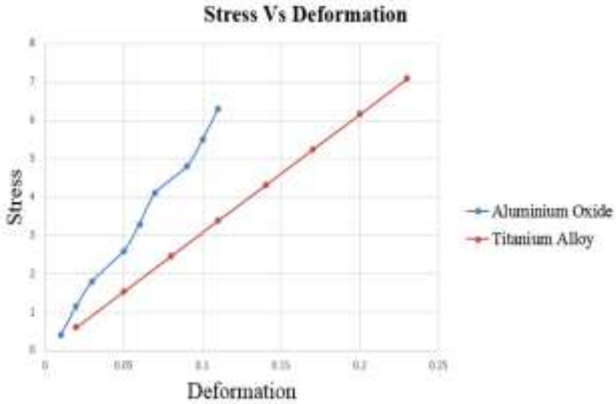


Figure: Shows the distribution of total deformation during standing



Graph: Stress Vs Deformation

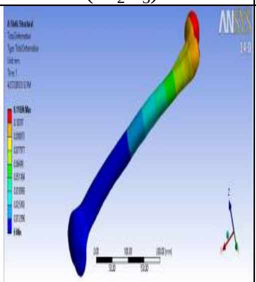
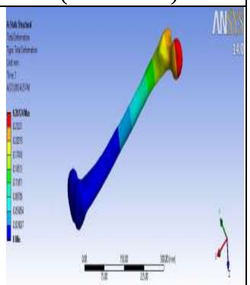
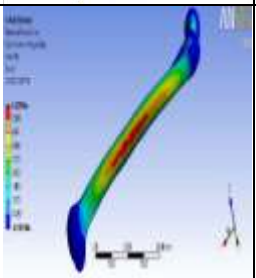
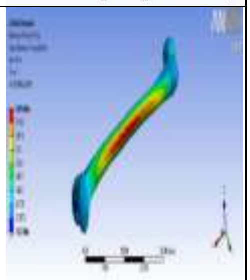
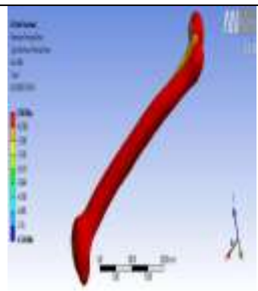
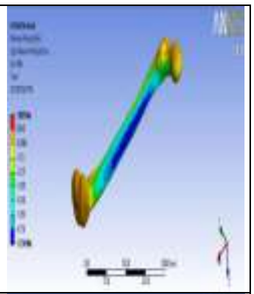
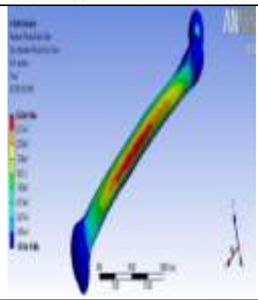
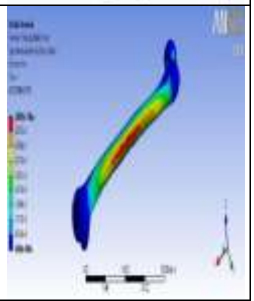
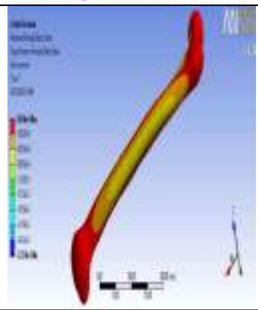
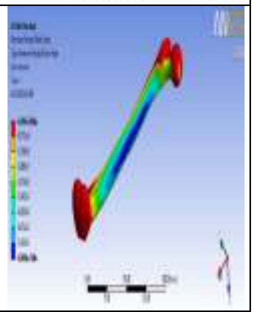
The above graph shows the details of stress and deformation of the femur bone during standing.



Graph: Comparison of Stress Vs Deformation for Titanium and Aluminium Oxide

The above graph shows the comparision of stress and deformation for aluminium oxide (Al_2 O_3) and titanium alloy (Ti-6Al-4V) under loading condition. From this we can say that titanium alloy absorbs more stress.

4.3 Comparison of Aluminium Oxide and Titanium Alloy

	Aluminium Oxide (Al_2O_3)	Titanium Alloy (Ti-4Al-6V)
Total Deformation		
Maximum Stress		
Minimum Stress		
Maximum Strain		
Minimum Strain		

5. Conclusion

It is observed that sudden accident and continuous vibration excitation is the main reason for femur bone failure. The results of this study show that the maximum chance of bone cracking is through bone shaft and neck region. The natural frequency and first ten mode shape of femur bone was determined using fixed-fixed boundary condition. The results of this study are verified by the experimental results available in literature ANSYS software has powerful analysis capabilities and CATIA V5 software has a powerful function of solid modelling.

They are suited for Finite Element Analysis of complex shapes. The 3D solid model is prepared by applying CATIA software and is transferred to ANSYS. In this research work we have considered the vibration problem of the femur bone using FEA method. Finite Element Analysis offers satisfactory results with additional ability to calculate regional mode and natural frequency with fracture locations during external loading condition.

6. Future Scope

Next work will be oriented to the experimental testing of the proposed solutions, in particular prototypes of the artificial Femur bone will be produced and mechanically tested according to ISO 7206 standard and data available in literature. Further, its biomedical requirements will be tested in terms of biocompatibility and osteointegration capability.

References

[1] Ajay Dhanopia[a,*], Prof. (Dr.) Manish Bhargava[b], 1877-7058 © 2017 the Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license.
[2] Ashwani Kumar[a], Himanshu jaiswal[a], Tarun Garg[a], Pravin P Patil[b], 2211-8128 © 2014 Elsevier Ltd. This is an open access article under the CC BY-NC-ND license.
[3] Sandeep Kumar Parashar, Jai Kumar Sharma* 2213-0209/© 2016 Published by Elsevier GmbH. This is an open access article under the CC BY-NC-ND license.
[4] Andrew E. Anderson, Benjamin J. Ellis, Steve A. Maas, and Jeffrey A. Weiss* How accurately can we predict the fracture load of the proximal femur using finite element models. Clin. Biomech. 1-8
[5] Farah Hamandi and Tarun Goswami, Whole-body vibration and low back pain: a systematic, critical review of the epidemiological literature 1992-1999, Int Arch Occup Environ Health 73,290-297.
[6] Royi Fedida, Zohar Yosibash, Charles Milgrom and Leo Joskowicz, A survey of finite element analysis in orthopedic biomechanics: The first decade. Journal of Biomechanics 16,385-409.
[7] Basafa, E., Armiger, R.S., Kutzer, M.D., Belkoff, S.M., Mears, S.C., Armand, M., 2013. Patient-specific finite element modelling for femoral bone augmentation. Med. Eng. Phys. 35, 860—865.
[8] Huiskes, R., Chao, E.Y.S., 1983. A survey of finite element analysis in orthopedic biomechanics: the first decade. J. Biomech. 16, 385—409.
[9] Khalil T. B., D. C. Viano and L. A. Taber, 1981. Vibrational characteristics of the Embalmed human femur. Journal of Sound and Vibration 75 417-436.
[10] Meng, L., Zhang, Y., Lu, Y., 2013. Three-dimensional finite element analysis of mini-external fixation and Kirschner wire internal fixation in Bennett fracture treatment. Orthop. Traumatol. Surg. Res. 99, 21—29,
[11] Basu, P.K., Beall, A.G., Simmons, D.J., Vannier, M., 1986. 3-D femoral stress analysis using CT scans and p-version FEM. Biomater. Med. Devices Artif. Organs 13, 163—186.

Author Profile

Nagaraja S N[1] received the M.Tech. Degree in Mechanical Engineering from M S Ramaiah Institute of Technology in 2014 and B.E in Mechanical Engineering from Sir. M.Vishveswaraya Institute of Technology in 2011, currently working as a Asst.Professor in the Dept. Of Mechanical Engg. at Kuppam Engineering college.