ELK ASIA PACIFIC JOURNAL OF MECHANICAL ENGINEERING AND RESEARCH- SPECIAL ISSUE
ISSN 2349-9368(Online);EAPJMER/ISSN.2454-2962/2016;

ELK
Asia Pacific Journals

National Conference on Futuristics in Mechanical Engineering
Madan Mohan Malaviya University of Technology

A REVIEW PAPER ON STRESS ANALYSIS OF FEMUR BONE

Ayatullah Ansari	Ram Bilas Prasad
Department of Mechanical Engineering, Madan Mohan Malaviya University of Technology	Department of Mechanical Engineering, Madan Mohan Malaviya University of Technology

Abstract

The human femur bone is the longest and heaviest bone in the body. Its length is 26% of the height of person. The study of human bone is gaining importance in order to characterise the bone material and enahance the understanding of bone failure during different loading conditions. This paper presents a review on the stress analysis of femur bone. For the finite element analysis of bone, the accurate geometry of bone is required. Various modelling techniques like Solidworks have been beendisccused for the creation of geometry of the bone. A review on stress analysis of femur bone has been done considering different loading condition.

Keywords: *Stress Analysis, Femur, Bone, Solid works, Ansys.*

INTRODUCTION

In 1972, a new method is introduced to calculate the stress distribution in skeleton parts with the help of finite element method and this method is useful in every field like biomechanics, engineering mechanics, etc. Under the loading condition, the stresses and strain generated in the skeleton parts provides new interest in orthopedic related science. In classic mechanism, the stresses and strain can be calculated by mathematical tools but due to irregular structural properties, the mathematical tools can not give the accurate results of stress distribution in bone. The gap has been filled by finite element analysis and it has ability to solve the stresses in complex geometry, under different loading conditions and the behavior of materials [11].

The finite element analysis is also used to determine the stresses in prosthesis, implant, artificial joints and fracture fixation devices. This method is very helpful in designing implants and easily select the materials instead of making a prototype of implant.

Cancellous bone has a strongly heterogeneous material with microstructural features. This has a significant effect on its macroscopic mechanical properties. Bone–prosthesis systems are usually analyzed on the basis of classical continuum mechanics. However, such an analysis is only justified under the assumption that the continuum quantities (e.g. stress and strain) do not vary substantially over intrinsic microstructural dimensions.

The femur bone is a vital part of the body. The movements of the joints for standing, sitting, and walking are possible by human femur bone. The important constituents of femur bone are cortical, compact tissues and other small parts as shown in Figure-1.

Bone

Bones are the most essential part of the body. It provides a complete structure of a body and the material of bone is a type of natural composite. There are 206 bones in human skeleton and the femur bone is the longest bone compared to all bone. The function of the human skeleton is to provide support, gives shape to our bodies, provides protection to the other systems and organs of the body, to provide attachments for muscles, to produce movement and to produce red blood cells. The human skeleton are shown in Figure-2.

Bone Problems

The bone is a part which is subjected to different loading conditions during standing, running, walking and jumping. Sometimes, the bone are not much stronger due to some problems like polio, paralysis, etc. Therefore, the stresses generated due to the activities are should be in range. Facing accident is also a cause which breaks the bone and then it can be joint by plaster

The main objective of the present paper is to identify the stress distribution in femur bone under the applied load and boundary conditions. The effect of torque, moment and oblique loading in femur bone. The isotropic properties of material and the linearly elastic behavior of bone is considered.

Material of Bone & 3DModel of Human Femur Bone

Bone is a type of natural composite material having varying properties at all points. The material for bone have isotropic property which means having identical properties in all direction. This assumption was taken due to its non-uniform cross-section throughout the length.

The 3D model of femur bone is created on Solid works software. The model of fractured bone and bone plates are also created in same software. The fractured bone is fastened with the help of bone plates by using screw which is also created in SOLIDWORKS. Providing bonded joint in between fractured human femur bone and frictional joint in between screw and plates. The different types of materials are used like stainless steel, nylon, titanium, alumina and PMMA to get the suitable material and found titanium as the best compared to others [3].

The modelling of femur bone in 3D animated blender software and converted a polygon shape into a 3D model. Imported this model into Ansys software for further analysis. The mechanical stress analysis is done in Ansys software [10].

Implant and its Materials

The implant in bone is introduced to increase the strength of the bone under the load condition. Sometimes, the artificial body parts are used in place of defected parts. The 3 mandibular parts with different superstructure implant are modeled on Ansys software and performed stress analysis on mandibular parts. The 3 models was first, 3 implants supporting three connected crowns, second 2 implants supporting a cantilever prosthesis and third model are 2 implants supporting a conventional fixed partial denture. These models are subjected to axial (AX), buccolingual (BL), or mesiodistal (MD) loads. The material of the implant are titanium, gold alloy and porcelain and their suitable properties are used in this research [5]. The 3-implant parts model have been generated, two implant-abutment are modeled with dimensions of regular platform (RP) group where a regular 4.3 mm diameter of abutment is connected to regular 4.3 mm diameter of implant, and a narrow platform (NP) group where a 3.5-mm diameter abutment is connected to a 3.5-mm diameter implant is used. The materials of the implant, abutment and screw are titanium and a restoration of Ni-Cr is designed over abutment [6].

Effects of Implant Positioning

The 3D model of mandible has been obtained by Computer Tomography (CT) scan and Morse tapper abutment and implant are modelled in SOLIDWORKS. The positioning of implant affected the stresses in implants and magnitude of the volume. The positioning of implants above crest level generated higher stress than placing it below crest level. If the stress location is known then implant volume should be increased according to the problem [7].

Loading Conditions of Bone

The most important factor while design or checking the strength of a bone or implant is the loading condition. The load acted on femoral head of human femur bone are compressive force and a torque of 10000 N-mm are also acted in case of walking, jumping or running [2].

A 3-D model of two implant-abutment with the prosthesis embedded on it are subjected to vertical loading and oblique (angle 35.6°) loading conditions. From analysis, it is found that stress generation in case of vertical loading is lower in comparison to oblique loading [6].In case of dental implant, the loading condition play the important role for finding the region where the stresses and deformation are maximum. The three loading conditions used are horizontal, vertical and oblique loading at 30°, 45° and 60°. The results obtained that the stresses is maximum in case of horizontal loading compare to oblique loading and minimum in case of vertical loading condition. Therefore, the implant neck is the weakest point in dental implant [12].

Finite Element Analysis of Bone

The FEA is a reliable and time-consuming tool which is used for stress analysis. The 3D model has been created in different softwares like SOLIDWORKS, MIMICS, PRO-E etc. and imported in ANSYS software for stress analysis. The materials are given by Mimis or

Ansys software and load and boundary condition are also applied in Ansys [1-7].

Mechanical Testing of Bone

The mechanical testing is performed on bone to validate the stresses of bone with the stresses obtained from finite element analysis. The mechanical testing methods are bending, impact loading and torsion these methods are used to test the behavior of material of bone. There are 3-point and 4-point bending tests in which 3-point is better to calculate the bending stresses in comparision to 4-point bending test due to uneven contour of bone, hard to contact the bone symmetrically throught out the length [8].

CONCLUSIONS

The human femur bone is also known as thigh bone:

[1] The material of bone have different properties at different points in the same bone. So, for performing stress analysis, it has to be assumed that the bone have isotropic properties which means having same property in all direction and it is linearly elastic.

[2] The 3D model of femur bone can be created in SOLIDWOKS or it can be converted from 2D CT scan file to 3D model. The material can be assigned easily in Ansys or Mimics.

[3] If some failure occurs in bone then prosthesis and implant are used to strengthen the bone and join two fractured part by using bone plates. The prosthesis (artificial) parts like leg, heart, etc. can be used to provide body parts who has not that parts.

[4] By using finite element analysis it is easy to calculate the stress distribution and deformations in bone and implants in bio-fields.

[5]

echanical testing like 3-point bending and 4-point bending can be performed to check the strength of the bone or to validate the result obtained from finite element analysis. The 3-point bending test can be easilyperformed.

References

[1] Uzair N. Mughal, "Finite element analysis of human femur bone", Int. Jnl. of Multiphysics, Volume 9, Number 2 (2015), pp 101-108.

[2] Nithin Kumar KC, "Biomechanical stress analysis of a Human Femur bone using ANSYS", Materials Today: Proceedings 2, 2015, pp- 2115 – 2120.

[3] P. S. R. S. Maharaj, "Numerical Analysis of Fractured Femur Bone with Prosthetic Bone Plates", International conference on design and manufacturing, Procedia Engineering 64, pp-1242-1251, 2013.

[4] TZuylan, "An Analysis of Anatolian Human Femur Anthropometry", Turk J Med Sci 32, pp-231-235© T.BUTAK, 2002.

[5] R. Stegaroiu, "Influence of Restoration Type on Stress Distribution in Bone Around Implants: A Three-Dimensional Finite Element Analysis", International Journal of Oral &

Maxillofacial Implants, Volume 13, No. 1, 1998.

[6] H. M. Abraham, "The effect of loading condition on peri-implant bone stress in regular and narrow diameter implants: A three-dimensional finite element analysis", Drug Invention Today, Vol 10, Issue 7, 2018, pp 1243-1245.

[7] J. P. Macedo, "Finite element analysis of peri-implant bone volume affected by stresses around Morse taper implants: effects of implant positioning to the bone crest", Computer method in Biomechanics and Biomedical Engineering, 2018, VOL. 21, NO. 12, ISSN: 1025-5842, pp-655-662.

A. Sharir, M. M. Barak, "Whole Bone Mechanics and Mechanical Testing", The veterinary journal 177, 2008, pp-8–17.

[8] Ning Xu, Xiaojian Y, "3D Artificial Bones for Bone Repair Prepared by Computed Tomography-Guided Fused Deposition Modeling for Bone Repair", dx.doi.org/10.1021/am502716t| ACS Applied Mater. interfaces 2014, pp-14952−14963.

[9] Moh. S. Masood, "Unconventional Modeling and Stress Analysis of Femur Bone under Different Boundary Condition", International journal of scientific & engineering research, Issue 12, December-2013, volume 4, ISSN 2229-5518, pp-293-296.

[10]

. Huiskes, "A survey of finite element analysis in orthopedic biomechanics: the first decade", 1983, biomechanics vol. 16. No. 6, pp-385-409.

[11]

sin-ChungCheng, "Influence of Deformation and Stress between Bone and Implant from Various Bite Forces by Numerical Simulation Analysis", Hindawi BioMed Research International Volume 2017, Article ID 2827953, 7 pages.

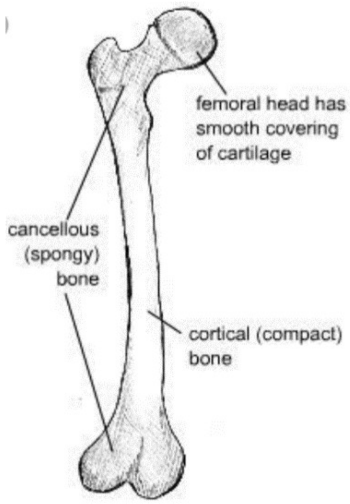

Figure 1: *Femur Bone of Human body [1]*

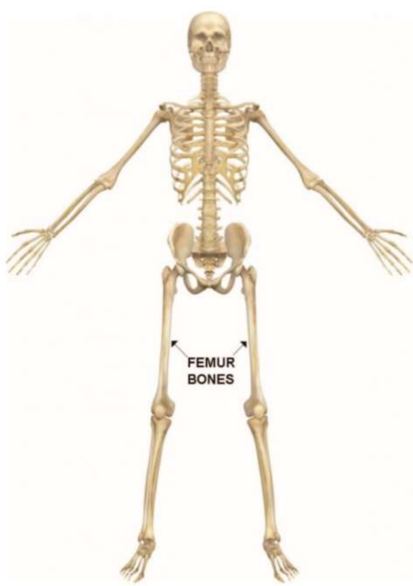

Figure 2: *The human skeleton [1]*

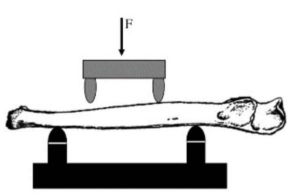

Figure 3:4-*point bending of a whole bone [8]*