International Conference On Emerging Trends in Engineering and Management Research
NGSPM's Brahma Valley College of Engineering & Research Institute, Anjaneri, Nashik(MS) (ICETEMR-16)
23rd March 2016, www.conferenceworld.in ISBN: 978-81-932074-7-5

A REVIEW PAPER ON BIOMECHANICAL ANALYSIS OF HUMAN FEMUR

K.S.Zakiuddin[1], I.A.Khan[2], Roshni A. Hinge[3]

[1]Prof. & Head, [2]Associate Prof., [3]PG Research Scholar,Dept. of Mechanical Engg.,

Priyadarshini College of Engineering, Nagpur, MH (India)

ABSTRACT

Femur bone is one of the most commonly fractured bones of human body, especially for the elderly people. It is known as the largest and longest bone in human body. It bears most of the body weight during activities such as standing, walking, and running. The main objective of this review paper is to analyze the behavior of human femur subjected to various forces and conditions. Forces normally experienced by humans during daily living activities, also in uncertain cases like accident, twist, etc causing femur deformation or failure. Thus, it is necessary to analyze the material properties, structure, load resistance and chance of failure of human femur. The present study includes the description of the structure and mechanical properties of the cortical and cancellous bone of the femur, analysis of joint and muscles forces acting on the femur, finite element analysis, vibrational behavior of the human femur, the experimental methods, compression, tension, bending and torsion tests resulting the strength of the femur.

Keywords: Biomechanics, Human Femur, FEA, Material Properties, Stress Analysis.

I. INTRODUCTION

Biomechanics is the development extension and application of mechanics for the purpose of understanding better the influence of mechanical loads on the structure, properties and function of living things. Biomechanics focuses on design and analysis, each of which is foundation of engineering[1].

Extending from the hip to the knee is femur. It is the largest, longest and strongest bone of the human skeleton. Its head fits into a pelvis socket called the acetabulum to form hip joint. The femur head is joined to the bone shaft by a narrow piece of bone known as the neck of the femur. The lower end of femur hinges with the tibia to form the knee joint. At the lower end, the bone is enlarged to form two lumps called the condyle that distribute the weight-bearing load on the knee joint. On the outer sides of the upper end of the femur are protuberance called greater trochanter and lesser trochanter. Macroscopically structure of femur consists of two types: cortical or compact bone which is a dense outer layer mainly resists bending, Cancellous or spongy or trabecular bone present in the interior of mature bones[2].

The neck of the femur is a point of structural weakness and a common fracture site in elderly people, especially in women suffering from osteoporosis and is usually associated with a fall and with age of 65 or above. This causes severe pain in the hip and legs cannot bear any weight. Occasionally, the broken ends of the bone become impacted i.e. wedged together, thus lesser pain and makes walking still possible but delay reports fracture and injury. Fracture of the shaft of the femur occurs when subjected to extreme force such as in a road traffic accident.

II. BIOMECHANICAL ANALYSIS OF HUMAN FEMUR BONE

Pradosh P. Dash, had been developed three dimensional finite element model of human femur with the help of computerized tomography scan data of the left human femur and the dynamic stress analysis has been carried out using commercial finite element software (ANSYS 14.0). The model is developed using two different material properties (isotropic & orthotropic) and the load have been taken during the daily activities (normal walking, standing up, stair climbing and knee bending). To check the efficacy of the present developed model the results are compared with those published literature[3]. A.Latif Aghili, was constructed three dimensional model of human femur by means of reverse engineering method and this three-dimensional finite element model of human femur has been analyzed under single, expanded and partial expanded loads. The material is assumed to have isotropic elastic characteristics. The results indicated that the important stress occurred at the inferior root of the femoral neck but maximum stress obtained at the femoral shaft [4].

Raza hedayati, investigates the impact of the geometry of the window perforated in the shaft part of this bone on its strength. Four window geometries, including square, circle, trapezoid and triangle were employed in order to assess yield stress under tension, compressive 3-point bending, 4-point bending and torsional loadings. Trapezoidal window showed much better resistance in 3-point bending and axial loadings compared to other window geometries, while it showed the weaker performance in torsional and 4-point bending loads. However, the femur bone is very unlikely to be loaded in 4-point bending. Moreover, in torsion, the femur bone with trapezoidal window was only 12% weaker than the femur bone with circular window (in axial loading, the femur bone with trapezoidal window had 33.6% higher strength than the bone with circular window)[5].

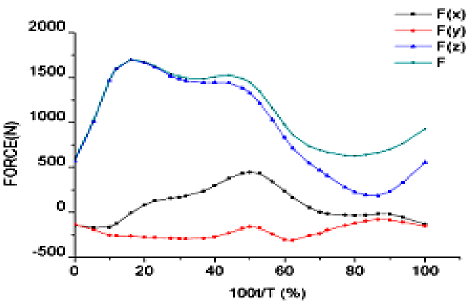

Fig. 1 Hip contact force during normal walking[3]

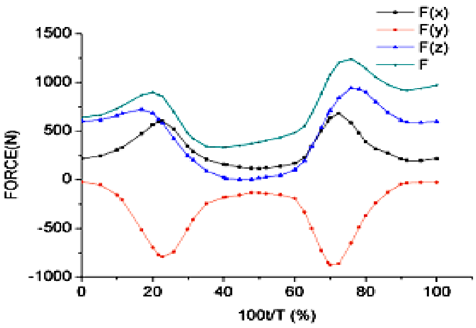

Fig. 2 Hip contact force during standing up[3]

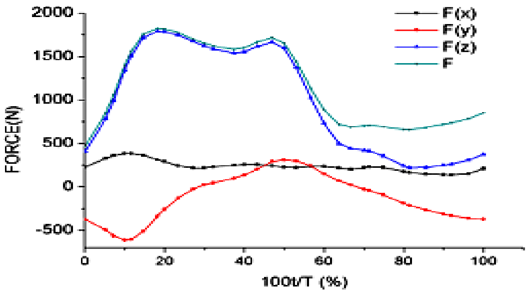

Fig. 3 Hip contact force during stair climbimg[3]

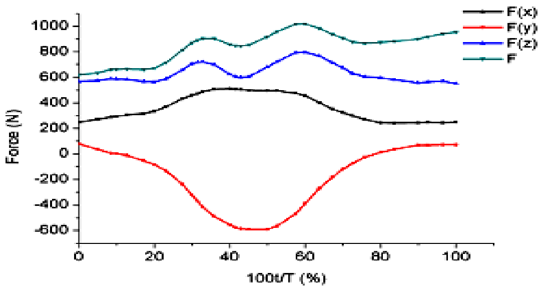

Fig. 4 Hip contact force during knee bending[3]

The three dimensional finite element model of the human femur had been constructed by means of the reverse engineering method and unlike many previous works, in this issue had been used full model instead of partial or half model and this three dimensional finite element model has been analyzed under single, expanded and partial expanded loads. The material is assumed to have isotropic elastic characteristics. The results indicated, there are two region for the maximum stresses and this kind of finite element model and its simplification has been verified with the published experimental results and this study has shown that the real produced stress-strain distribution at the human femur when a person was in the single-leg stance phase of normal running or jumping for the first time [6]. A three dimensional model of the human femur bone has been developed and the data associated with the hip contact forces for normal walking and standing up during one cycle has been employed on the femur bone in order to investigate behavior of the femur bone during these activities. The finite element results (stresses) are obtained and compared with previous studies. The behavior of the stresses that obtained in the present study is similar to those found in the literature [7].

Amrita Francis, had been created 3-D finite element models of the right human proximal femur for three male patients of 17 yrs, 32 yrs and 40 yrs using CT scan data for FEA loaded by individual body weight of 75 Kg, 72 Kg and 66 Kg respectively which is shared equally by the lower limbs, at different inclination angles and to determine the total deformation, equivalent Von Mises stress, maximum principal stress, fatigue tool and percentage variation[8].

Table I Allowable forces for each loading [5]

	Tension	Compression	3P Bending	4P Bending	Torsion
Square	260 N	260 N	118 N	200 N	8.7 N.m
Circle	398.5 N	398.5 N	113.5 N	195.5 N	9.32 N.m
Triangle	359.5 N	359.5 N	91.5 N	160.5 N	6.94 N.m
Trapezoid	532.5 N	532.5 N	152.5	131.5 N	8.125 N.m

The aim of Nawapol Saktaveekulkit was to evaluate the mechanical performance of gamma 3 long nail in thai femoral bone. A Three-dimensional CAD model of the nail was created from Computed Tomography (CT) scan, and then the nail was inserted in to the bone, by Virtual Simulation, into the intramedullary canal. Then the geometric mismatch between the nail and intramedullary canal were measured. Finally, the mesh model of the bone and the devices was created for finite elements analysis. The strain on the femur and the stress distributed on the screw and nails were observed under walking conditions. The result showed that the maximum equivalent total strain occurred in the middle gap fracture because the narrowest cross section area where the load sharing was more than the other region [9].

Table II Illustrate the parameters required for the assignment of material properties for FEA [7]

Parameter	Cortical bone	Trabecular bone
Hounsfield Unit (HU)	2200	800
Density(g/cm^3)	2.0208	1.3712
Modulus of Elasticity(MPa)	E_1= 6982.9	E_1= 2029.4
	E_2= 6982.9	E_2=2029.4
	E_3= 18155	E_3= 3195.3
Poisson's Ratio	υ_{12}=0.4	υ_{12}=0.4
	υ_{23}= 0.25	υ_{23}= 0.25
	υ_{31}= 0.25	υ_{31}= 0.25
Shear Modulus (GPa)[16]	G_{12}= 4.69	G_{12}= 4.69
	G_{23}= 5.61	G_{23}= 5.61
	G_{31}= 7.68	G_{31}= 7.68

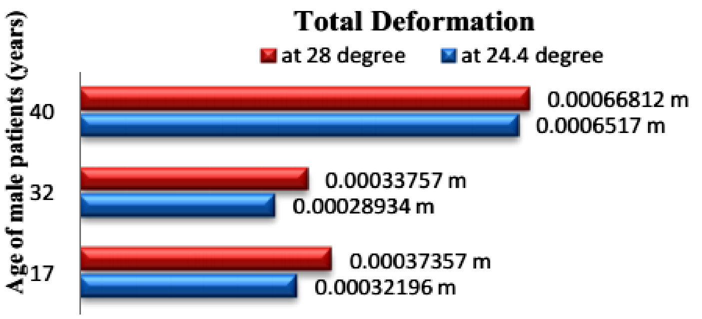

Fig.5 Bar chart of total deformation vs. age of male patients (years)[8]

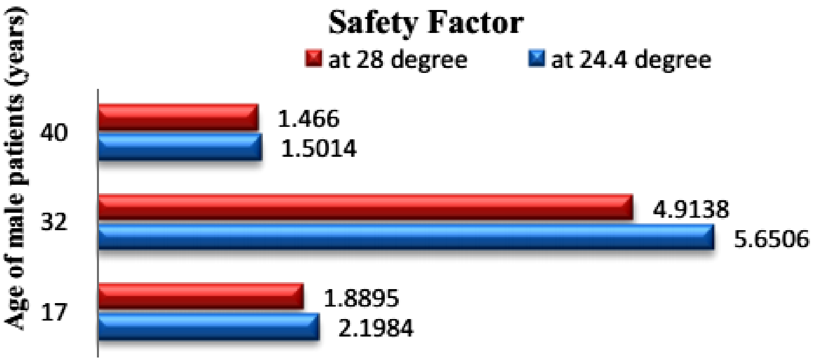

Fig.6 Bar chart on safety factor vs. age of male patients (years)[8]

A finite element model of human femur bone with accurate geometry and stainless steel biocompatible material properties is developed from CT scan data. The main aim was to construct a model of real proximal femur bone for evaluating the finite element analysis. This would helpful for realistic investigations on the mechanical behavior of bone structures. The behavior of the bone is analyzed in ANSYS under physiological load conditions of patients. The obtained stresses are useful for the tests on strength, fixation and fixture of customized implants.[10]. Biomechanical behaviors of five different configurations of screws used for stabilization of femoral neck fracture under axial loading have been examined and which configuration is best has been investigated. Femoral neck fracture was modeled by Solid Works software for five different configurations: dual parallel, triple parallel, triangle, inverted triangle and square and computer-aided numerical analysis of different configurations were carried out by ANSYS Workbench finite element analysis (FEA) software. For each configuration, mesh process, loading status (axial), boundary conditions and material model were applied in finite element analysis software. Von Mises stress values in the upper and lower proximity of the femur and screws were calculated. According to FEA results, it was particularly advantageous to use the fixation type of triangle configuration. The lowest values are found as 223.32 MPa at the lower, 63.34 MPa at the upper proximity and 493.24 MPa at the screws in triangle configuration. This showed that this configuration creates minimum stress at the upper and lower proximity of the fracture line[11].

The aim of M.S.Kulkarni was, evaluating material properties of femur bone, so as to facilitate further study of total hip joint and replacement of joint. These properties are required to be determined before F.E.M. analysis of indegenised hip joint to study its stability in the bone. This experimental study aims at determining orthotropic behavior of cancelleous portion of cadeveric femur bone. This property may prove useful in studies related to total hip joint replacements. This study attempts at providing comprehensive items of mechanical properties of cadeveric cancellous distal femur, through series of mechanical tests, which comprised of tensile testing, compression testing, shear testing[12]. An unconventional approach is employed for modeling of complicated geometry of human femur bone to make realistic investigations. Two orthogonal views are employed for modeling of complex femur bone with the help of 3D animated open source software Blender 2.63a. Pro/ENGINEER translated Blender prepared polygonal model into CAD model that is imported to ANSYS for analysis under different boundary condition. Stress and deformation distribution varies with boundary condition. It has observed that Maximum stresses and minimum deformations are located at restraint end femur bone which indicates cantilever beam behavior of femur bone [13].

Femur 3D finite-element (FE) analysis based on CT data requires both accurate geometry representation and mechanical properties assignment. A "structure based" method is suggested for the reconstruction of a FE model so that the geometry is represented by smooth surfaces extracted from the CT data including a separating surface between cortical and trabecular regions. Then, p-FE auto-mesh is generated. It was a first step towards a p-FEM

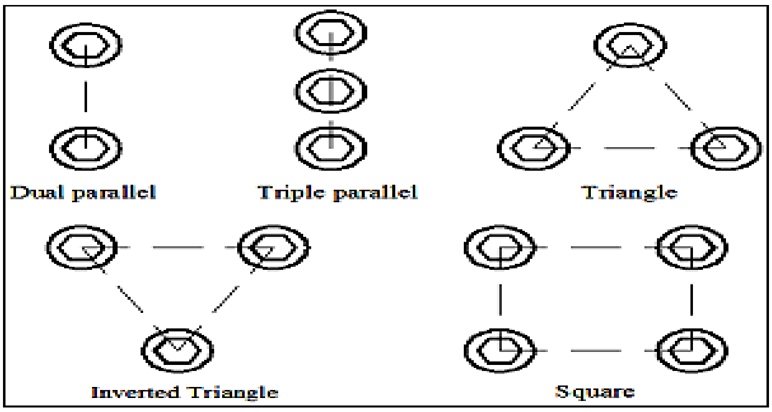

Fig.7 Schematic view of five different configurations of femoral neck fracture [11]

simulation of human femurs based on CT data in order to optimize a protocol for cervical fracture surgery fixation[14]. Analyze statistically results in biomechanical testing of fixation of femoral neck Pauwels type III fractures, on synthetic bone, with dynamic condylar screw (DCS) and control group. Methods: Ten synthetic bones of a national brand were used. Test Group: fixation was performed after osteotomy at 70° tilt using DCS plate with four holes. We analyzed the resistance of this fixation with 5 mm displacement and rotational deviation (Step 1) and with 10 mm (Step 2). Control group: the models were tested in their integrity until the femoral neck fracture occurred. The result shows that there is no significant difference between the DCS boards and the control group exposed to full resistance [15].

Dmitrij B. KAREV had been studied the stressed state peculiarities of cortical and trabecular bones by two-point asymmetric screw fixation with implant for femoral neck fracture. Layer construction mechanic methods are used for analysis of stresses in cortical and trabecular bones. Biomechanical conditions for non-opening of the junction of the bone parts being joined are determined. It has been found that the total tightness of the broken parts when they rest against each other is secured over the whole fracture section without junction opening under condition that fixing screws are positioned in the trabecular bone without penetration of the thread side surface into cortical bone. It has been found that solid tightness of the broken parts against each other is secured over the whole fracture section without junction opening under condition that fixing screws are positioned in the trabecular layer without penetration of the thread side surface into cortical layer [16].

R.M.Sherekar was presenting the femur bones analysis. The boundary conditions are applied here, generally producing excessive femoral deformation, and although it has been shown that the muscle/tendon forces influencing femoral deflections and dynamic loadings. It is hypothesize that careful application of physiologically based constraints can produce physiological deformation this causes straining, of the femur. Five boundary condition cases were applied to a finite element model of the standardized femur. The aim of the work was to present the influence of different mechanical properties of these three bones and comparative study on the obtained results[17].

III. VIBRATIONAL ANALYSIS OF FEMUR

Vibration analysis of femur bone had been studied by the help of finite element simulation to provide more insight in designing bio-aided equipments or protective sports equipments for femur. The simulations were performed using the open source software – Elmer. The vibration patterns for first twenty modes were studied. The vibration analysis show that the natural frequency of vibration varies from 964 Hz to 10.8 kHz. The external excitation on the femur bone must be avoided to coincide with these natural frequencies; otherwise it could lead to fracture of the bone. Mode shapes are very important in evaluating the behavior of any structure to an external response. In this work, FEA of femur bone was performed using a software Elmer. The natural frequencies and mode shapes for femur bone were identified for fixed-fixed boundary condition[18].

Table III Mechanical properties of bone in different phases[17]

Bone	Living Bone	Dead Bone
	Young modulus of bone, MPa	
Femur	17 260	20 202
Tibia	19 040	20 590
Fibula	18 540	21 080

Table IV Mode number and natural frequency[18]

Mode Number	Natural Frequency (Hz)
1	946.2363
2	1077.009
3	2043.75
4	2348.073
5	3523.645
6	4109.738
7	4825.301
8	5111.147
9	5423.284
10	6009.456
11	6898.864
12	7106.023
13	7343.545
14	7770.327
15	8444.315
16	8911.491
17	9365
18	9968.934
19	10300.25
20	10792.92

IV. CONCLUSION

Numerous investigations has been done on the human femur like physical test yielding knowledge on the bone properties, digitized and modelled in many different finite element programs even numerically and mathematically both at the tissue level and at macroscopic level. Much work has been done to analyse the material properties, structure, load resistance and chances of failure of human femur by methods ranging from mechanical and acoustic testing to more theoretical means. In this work, finite element analysis had been performed on human femur using ANSYS software while in vibrational analysis of femur same is performed using a software Elmer. The problems and issues faced by the orthopaedic surgeons during hip implant have been solved by these investigations. All these investigations may also help for clinical diagnosis of osteoporosis, physical therapy, better understanding of bone remodelling process essential for orthopaedic surgeon in femur surgeries and bone prosthesis.

REFERENCES

[1] A.Francis, A.Shrivastava,C. Masih,N.Dwivedi, P.Tiwari, R.Nareliya and V.Kumar,"Biomechanical analysis of human femur: a review", Journal of Biomedical and Bioengineering , Volume 3, 2012.

[2] Nather Aziz, Ong H.J.C.and Aziz Zameer, Structure of bone, 5.

[3] Pradosh Pritam Dash, Kaushal Kishor , S. K. Panda,"Biomechanical stress analysis of human femur bone", International Conference of Computer Aided Engineering(CAE 2013).

[4] A. Latif Aghili, A.M. Goudarzi, M.H. Hojjati,S.M. Rabiee, Misagh Imani and Ali Paknahad," Biomechanical Investigation of Human Femur by ReverseEngineering as a Robust Method and Applied Simplifications", World Applied Sciences Journal 28(12), 2013.

[5] Reza Hedayati, Seyyed Mohammadreza Fallah Alipour Zare," Effect of Window Geometry on Strength of Shaft Part of Femur Bone", Periodica Polytechnica Mechanical Engineering 59(2), pp. 81-87, 2015.

[6] A. Latif Aghil, A. Moazemi Goudarzi, Ali Paknahad, Misagh Imani and A. Abouei Mehrizi," Finite Element Analysis of Human Femur by Reverse Engineering Modeling Method", Indian Journal of Science and Technology,Vol 8(13), 47884, July 2015.

[7] A.E. Yousif , M.Y. Aziz," Biomechanical Analysis of the human femur bone during normal walking and standing up", IOSR Journal of Engineering (IOSRJEN) Volume 2, PP 13-19,2012.

[8] Amrita Francis,Veerendra Kumar," Computational Modeling of Human Femur using CT Data for Finite Element Analysis", International Journal of Engineering Research & Technology (IJERT)Vol. 1 , 2012.

[9] Nawapol Saktaveekulkit, Panya Aroonjarattham, Kriskrai Sitthiseripratip, Banchong Mahaisavariya and Kitti Aroonjarattham," Biomechanical Study of Thai Femoral Bone Fracture with Gamma 3 Long Nail: Finite Element Analysis",The Second TSME International Conference on Mechanical Engineering, 19-21, 2011.

[10] Rahul M. Sherekar, Sachin V. Bhalerao," Biomechanical & FE Analysis of The Customized Femur Implant For Performance Evaluation ",International Journal of Research In Science & Engineering, Vol 1, Issue 3.

[11] K. Gok and S. Inal," Biomechanical comparison using finite element analysis of different screw configurations in the fixation of femoral neck fractures", Mechanical Sciences, 6, 173–179, 2015.

[12] Mrudula S. Kulkarni, S.R. Sathe," Experimental Determination of Material Properties of Cortical Cadeveric Femur Bone", Trends Biomater. Artif. Organs, vol 22(1),pp 9-15,2008.

[13] Muhammad Shahzad Masood, Atique Ahmad, Rizwan Alim Mufti," Unconventional Modeling and Stress Analysis of Femur Bone under Different Boundary Condition", International Journal of Scientific & Engineering Research, Vol 4, Issue 12, 2013.

[14] Royi Fedida, Zohar Yosibash, Charles Milgrom and Leo Joskowicz,"Femur Mechanical Simulation Using High-order FE Analysis With Continuous Mechanical Properties", 2nd International Conference on Computational Bioengineering ,Lisbon, Portugal, 2005.

[15] Anderson Freitas, Rafael Almeida Maciel, Renato De Almeida Lima,"Mechanical Analysis of femoral Neck Fracture Fixation With Dynamic Condylar Screw in Synthetic Bone", Acta Ortop Bras. 2014.

[16] Dmitrij B. KAREV, Vladimir G. BARSUKOV,"Biomechanical Analysis of Two-point Asymmetric Screw Fixation With Implant For Femoral Neck Fracture",2012.

[17] Sherekar R.M, Pawar A.N.," Numerical Analysis of Human Femoral Bone in Different Phases", International Journal of Engineering and Innovative Technology (IJEIT)Vol 2, Issue 12, 2013.

[18] A. Gupta, K.M.Tse,"Vibrational Analysis of Femur Bone Using Elmer", Journal of Engineering Science and Technology, 2014.