Journal of Biomedical and Bioengineering
ISSN: 0976-8084 & E-ISSN: 0976-8092, Volume 3, Issue 1, 2012, pp.-57-62.
Available online at http://www.bioinfo.in/contents.php?id=87

FINITE ELEMENT APPLICATION TO FEMUR BONE: A REVIEW

RAJI NARELIYA[1] AND VEERENDRA KUMAR[2]

[1]Hitkarni College of Engineering and Technology, Jabalpur, M.P., India.
[2]Government Engineering College, Jabalpur, M.P., India.
*Corresponding Author: Email- raji_nareliya@yahoo.co.in and prof.veerendra.kumar@gmail.com

Received: April 30, 2012; Accepted: June 04, 2012

Abstract- Biomechanics is the study of the structure and function of biological systems by means of the methods of mechanics. Finite element analysis is a computer based numerical analysis method which can be used to calculate the response of a model to a set of well-defined boundary conditions. Unlike conventional analysis methods, FE analysis can be used to analyze structures of complicated geometry and inhomogeneous material properties. This nature of FE analysis makes it an ideal tool for analyzing living tissue such as bone, which has complicated geometric shapes and inhomogeneous material properties. The FE method can be briefly explained as follows. The objective of an FE analysis is to find the distribution of an unknown within a body. For the purpose of this explanation, assume that the unknown is displacement. The body is divided into small parts of simple shapes that are called elements. Elements are interconnected at points called nodes. The displacement is assumed to vary over each element in a predefined manner. This variation is usually defined by a polynomial. The polynomial, the shape of the element, and the number of nodes per element are not entirely independent of each other. The governing equations relating force to displacement for each element are formulated and assembled to give a collection of equations. These equations describe the behavior of the collection of elements and hence the behavior of the body. In the early stages, FEA method was used from an academic point of view rather than from a practical application point of view. Since then FE analysis has been used in orthopedics to analyze a variety of topics, such as bone remodeling, fracture healing, intramedullary nailing, and artificial joints. This review paper is an attempt to summarize the application of finite element analysis of human femur bone according to year wise.
Key words- Biomechanics, Modeling, FEA, FEM, Femur.

Citation: Raji Nareliya and Veerendra Kumar (2012) Finite Element Application to Femur Bone: A Review. Journal of Biomedical and Bioengineering, ISSN: 0976-8084 & E-ISSN: 0976-8092, Volume 3, Issue 1, pp.-57-62.

Introduction

Biomechanical Engineering is a bio-engineering sub discipline which applies principles of mechanical engineering to biological systems and stems from the scientific discipline of biomechanics. Finite element analysis (FEA) has many useful applications in the field of medical science to study the mechanical behavior of human organs like bones under static and dynamic loading conditions and to study injury mechanism, etc. Finite Element Analysis is a computer based numerical analysis which can be used to analyze structures of complicated geometry and inhomogeneous material properties. FE Method is widely accepted as an ideal tool for biomechanics modeling which has complicated shapes and heterogeneous material properties. Bone density modulus of elasticity and position of femur are influencing factors on load transfer mechanism and, if not entered correctly in finite element model, may produce unreliable results. For this reason, anatomically accurate finite element model of bones with accurate geometry and material properties retrieved from CT scan data are being widely used to make realistic investigations on the biomechanical behavior of bone structure and implant fixation. In clinical application of computational biomechanics, mechanical analysis considering only standard or normal patient models is not sufficient to discuss clinical problems. Analysis based on individual or patient specific modeling considering characteristics of organ shape and tissue construction is indispensable.

Bones are rigid organs that form part of the endoskeleton of vertebrates. They move, support, and protect the various organs of the body, produce red and white blood cells and store minerals. Bone tissue is a type of dense connective tissue. Bones come in a variety of shapes and have a complex internal and external structure, are lightweight yet strong and hard, and serve multiple functions. The primary tissue of bone, osseous tissue, is a relatively hard and lightweight composite material, formed mostly of calcium phosphate in the chemical arrangement termed Calcium hydroxyl apatite. It has relatively high compressive, of about 170 MPa but poor tensile strength of 104-121 MPa and very low shear stress strength (51.6 MPa). While bone is essentially brittle, it does have a significant degree of elasticity, contributed chiefly by collagen. There are five types of bones in the human body : long, short, flat, irregular and sesamoid [14].

Femur Bone

In human anatomy, the femur is the longest and largest bone. The average adult male femur is 48 centimeters (18.9 in) in length and 2.84 cm (1.12 in) in diameter at the mid shaft, and can support up to 30 times the weight of an adult. It forms part of the hip joint and part of the knee joint, which it is located above. In the erect posture it is not vertical, being separated above from its fellow by a considerable interval, which corresponds to the breadth of the pelvis, but inclining gradually downward and medial ward, so as to approach its fellow toward its lower part, for the purpose of bringing the knee-joint near the line of gravity of the body. The degree of this inclination varies in different persons, and is greater in the female than in the male, on account of the greater breadth of the pelvis. The femur, like other long bones, is divisible into a body and two extremities. Human femur has been capable of resisting compression forces of 800-1100 Kg. [16] There are four eminences the human femur: the head, the greater trochanter, the lesser trochanter, and the lower extremity. The upper extremity represent head, neck, greater and lesser trochanter, while lower extremity have lateral condyle medial condyle and patellar surface. The shaft of femur is cylindrical with a rough line on its posterior surface.

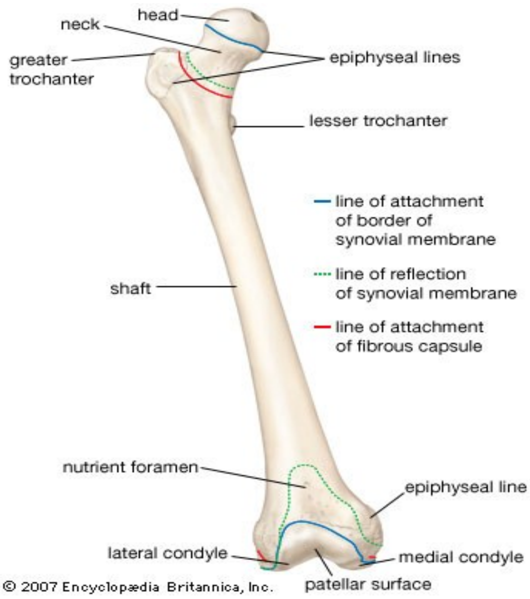

Fig. 1- Anatomy of Femur Bone

Architecture of femur: "The femur obeys the mechanical laws that govern other elastic bodies under stress; the relation between the computed internal stresses due to the load on the femur-head, and the internal structure of the different portions of the femur is in very close agreement with the theoretical relations that should exist between stress and structure for maximum economy and efficiency; laws of bone structure have been demonstrated for the femur are :

1. The inner structure and external form of human bone are closely adapted to the mechanical conditions existing at every point in the bone.
2. The inner architecture of normal bone is determined by definite and exact requirements of mathematical and mechanical laws to produce a maximum of strength with a minimum of material" [13].

The Joints of the Femur: Proximally (towards the hip) the femur articulates (attaches to) the acetabulum which is made up of all three hip bones: the ilium, ischium, and pubis. Distally (towards the foot) the femur articulates with the tibia and fibula.

The hip joint, scientifically referred to as the acetabulofemoral joint is the joint between the rounded head of the femur and the cuplike acetabulum of the pelvis. The hip joint is a ball-and-socket joint; the round head of the femur rests in a cavity (the acetabulum) that allows free rotation of the limb. It forms the primary connection between the bones of the lower limb and the axial skeleton of the trunk and pelvis. Both joint surfaces are covered with a strong but lubricated layer called articular hyaline cartilage. The cuplike acetabulum forms at the union of three pelvic bones, the ilium, pubis, and ischium. Its primary function is to support the weight of the body in both static and dynamic postures. The hip joints are the most important part in retaining balance.

The knee joint joins the thigh with the leg and consists of two articulations: one between the femur and tibia, and one between the femur and patella. It is the largest joint in the human body and is very complicated. The fibula is connected to the tibia and does not form part of the knee joint. The patella articulates with the patella femoral groove on the femur, but not with the tibia. Two condyles on the distal femur articulate with two condyles on the proximal tibia to form the tibio femoral joint. The radius of the condyle curvature in the sagittal plane becomes smaller toward the back. This diminishing radius produces a series of involutes midpoints.

This literature review presented in this paper for femur bone only. The attempt is made to arrange whole literature according to year wise.

Literature Review

A three-dimensional finite element models (FEM) of the human femur is developed from computed tomography data, with flexible and regular meshing 'brick' elements. Three different material properties were modeled for strain energy density (SED) patterns of the different femoral parts were most sensitive to the implementation of an iliotibial tract force [24]. A three-dimensional finite element models were created for an intact femur and a synthetic femur implanted with cement less prosthesis for investing proximal load transfer under two loading conditions, by using a coupled experimental and finite element analysis The approach was used to investigate a press-fitted and a fully bonded bone-prosthesis structure to identify the stem-bone behavior for both interface conditions and their implications for proximal bone load transfer [27]. Geometry-based and voxel-based finite element (FE) method preprocessor of the human femur compared with experimental validation by strain gauge measurements (principal stress) [23]. FE model of human femur was created by using CT image data just after hip joint arthroplasty. The modern CT scanners accurately reconstruct the cross-section geometry of titanium alloy hip joint prosthetic stems and a new measuring procedure dealing with the geometry of real stems was developed with accuracy. Secondly, the artifacts generated by a prosthetic stem on the radiological density of the bone were analyzed, and their effects on the assessment of FEM material properties were evaluated [47]. Ability test of DXA, QCT and FE techniques are used to predict fracture load non-invasively, in a simple load configuration which produces predominantly femoral neck fractures by configuring load at the single stance phase portion of normal gait using three dimensional FE techniques [10]. FE models of two right canine femurs with and without implants based on contiguous computed tomography (CT) scans were constructed for estimating subject-specific stress-shielding fewer than four different loading conditions and two bone density-modulus relationships for calculating stress. Stress-shielding was defined as the decrease of strain energy per gram bone mass in the femur with the implant in place relative to the intact femur. The analyses showed that the same loading condition and density-modulus relationship was used for both the four loading conditions and two bone density-modulus relationships the difference in stress-shielding between the two subjects was essentially constant [44]. A set of programs were created to investigate the geometrical and biomechanical aspects of pre-operative planning to automatically perform a simulation of intertrochanteric osteotomies on a three-dimensional finite element model of the human proximal femur based on computed tomography (CT) data and using uniform brick-shaped elements. To eliminate artifacts resulting from the rough surface of the brick elements, the femoral head was represented by a tetrahedron-based head that included a cartilage layer and a subchondral cortical zone [35]. A new finite element model (FEM) based on an elasto-plastic behavior of ultra high molecular weight polyethylene (UHMWPE) was used to study the wear behavior of UHMWPE acetabular cup, which has a 32 mm diameter femoral head. The model imposed a plastic yield stress of 8 MPa on the UHMWPE so that any stresses beyond this would automatically be redistributed to its neighbor. The FEM model adopted a unique mesh design based on an open cube concept which eliminated the problems of singularities. Wear prediction combined the influences of contact stress, sliding distance and a surface wear coefficient. The new model predicted significantly higher volumetric wear rate (57 mm^3/yr) well within the average reported clinical values. The model was also used to study the effect of friction and clearance between the acetabular cup and the femoral head [34]. A procedure for the generation of FE models of human bones was regenerated from data collected in vivo, robust, accurate, automatic and general enough to be used in orthopedic application. Robustness, automation and numerical accuracy of the proposed method were assessed on five femoral CT data sets of patients affected by various pathologies. The method was verified by processing a femur, an ileum, a phalanx, a proximal femur reconstruction, and the micro-CT of a small sample of spongy bone. The method was robust enough to cope with the variability of the five femurs, producing meshes with a numerical accuracy and a computational weight comparable to those found in vitro. Even when the method was used to process the other bones, the levels of mesh conditioning remained within acceptable limits [43]. A three dimensional finite element model of realistic femur (composite bone) and a femur with intramedullary was created to compare experimentally measured strains on the surface of the femur. FE model identify implant/bone load sharing patterns as well as areas of stress concentration in both the intramedullary nail and the bone, when statically locked by one or two screws at either end [5]. Idea of topology optimization was introduced in engineering to simulate bone morphology. The external shape of bone structure was predicted with the quantitative bone functional adaptation theory. The high order nonlinear equation of bone remodeling combining with the finite element method was adapted to a rectangular design domain, which occupies a larger space than the external shape of bone structure. The proximal femur was used here as an example, whose external shape and internal density distribution were simultaneously simulated and quantitatively to validate that the external shape of bone structure could be successfully predicted in this way. Authors provides computational basis for further studies on osteophyte formation, osteoporosis, osteoarthritis, bone growth and even bone evolution, etc. [45]. General procedures to automatically generate subject-specific finite element models of bones were used from CT data and estimate the accuracy of one real femur. This femur was tested in vitro under five different loading scenarios and the results of these tests were used to verify how the adoption of a simplified two material homogeneous model would change the accuracy with respect to the density-based inhomogeneous one, especially for the epiphysis and metaphysical proximal regions of the bone [37]. Effects of assumed isotropy was quantify, by comparing continuum-level voxel models of a healthy and a severely osteoporotic proximal femur with recently analyzed micro -FE models of the same bones. The micro-model element size was coarsened to generate continuum FE models with two different element sizes and two different density–modulus relationships used for wet and ash density. All FE models were subjected to the same boundary conditions that simulated a fall to the side, and the stress and strain distributions, model stiffness and yield load were compared. The smallest differences between the continuum-level model and micro-level model predictions of the stiffness and yield load were obtained with the coarsest element size [42]. Human proximal femur was simulated by high-order finite elements and the results were validated by experimental observations. A fresh-frozen human femur was scanned by quantitative computed tomography (QCT) and a quasi-static force sensitivity analyses were performed to quantify parameters that mostly influence the mechanical response after loading. Young moduli correlated to QCT Hounsfield Units by relations Comparison of CT scan-based finite element model predictions of proximal femoral fracture load and provide predictions that match the experimental results closely. [46] Trabecular bone was simulating to in the human proximal femur by using topology optimization and quantitatively investigated the validity of Wolff's law. Topology optimization iteratively distributes material in a design domain producing optimal layout. Authors used a two-dimensional micro-FE model with 50 mm pixel resolution to represent the full trabecular architecture in the proximal femur, and performed topology optimization to study the trabecular morphological changes under three loading cases in daily activities and the non-orthogonal intersections were constructed to support daily activity loadings in the sense of optimization, as opposed to Wolff's drawing [12]. Demonstrate the inclusion of subject-specific loading conditions drastically influences the calculated stress distribution, and correlate between calculated stress distributions as well as also changes in bone mineral density (BMD) after THR. For two patients who received cement less THR, personalized finite element (FE) models of the proximal femur were generated representing the pre and post-operative geometry. Average values of the Von Misses stresses were calculated for relevant zones of the proximal femur. Subsequently, the load cases were interchanged and the effect on the stress distribution was evaluated. Finally, the subject-specific stress distribution was correlated to the changes in BMD at 3 and 6 months after THR [20]. Mechanical and fluid flow scaffold properties are macroscopically derived by means of the homogenization technique while the variables at the microscopic level are obtained by invoking the localization problem. As a first approach, cell migration within the scaffold is macroscopically modelled as a diffusion process based on the Fick's law, with the diffusion coefficient depending on the size and spatial distribution of pores. At the microscopic scale, bone growth at the scaffold surface is considered to be proportional to the cell concentration and regulated by the local strain energy density. The mathematical model proposed has been numerically implemented using the finite element method (FEM) and the Voxel-FEM at the macroscopic and microscopic scales, respectively. The model has been qualitatively compared with experimental results found in the literature for a scaffold implanted in the femoral condyle of a rabbit achieving a good agreement [17]. The accuracy of a simple strain-based method was assessed for the prediction of failure in the femoral neck, through comparison with experimental fracture tests [2]. Quasi-static loading of human femur with β-spline based modeling was simulated and its 3D finite element analysis with graded element. β-spline surface representation method is extended to represent material composition to develop heterogeneous solid model of proximal femur. Lagrangian graded element is used to assign inhomogeneous isotropic elastic properties in finite element model to improve the performance. To test the feasibility of the model, sensitivity of simulation is investigated and validates the model; numerical results are compared with experimental work from another reference paper [31]. A three-dimensional model presented for bone remodeling taking into account the hierarchical structure of bone. The process of bone tissue adaptation is mathematically described with respect to functional demands, both mechanical and biological, to obtain the bone apparent density distribution and the trabecular structure. A three-dimensional model of the proximal femur illustrates the distribution of bone apparent density as well as micro structural designs characterizing both anisotropy and bone surface area density [6]. An oval method was used for generating high-order finite element (p-FE) models from CT scans and validated by experimental observations on two fresh frozen femurs. A fresh frozen femur of female was scanned under two different environments: in air and immersed in water and this proximal femur was quasi-statically loaded. The two QCT scans were manipulated to generate p-FE models that mimic the experimental conditions. In addition, the material assignment strategy was reinvestigated. The inhomogeneous Young's modulus was represented in the FE model using two different methods, directly extracted from the CT data and using continuous spatial functions. They compared p-FE displacements and strains of the wet CT model to the dry CT model and to the experimental results. Finally, the p-FE results of all three fresh frozen human femurs compare very well to experimental observations exemplifying that the presented method may be in a mature stage to be used in clinical computer-aided decision making [38]. Utilization of a principal strain fixation ratio was introduced which is defined as the ratio of principal strains that develop in a fixated bone relative to the principal strains that develop in the same bone in an intact state. The SR field output variable is theoretically independent of load amplitude and also has a direct clinical interpretation with SR representing stress shielding. A combined experimental and numerical study was performed with cadaveric proximal femora intact and following fracture fixation to quantify the performance of the SR variable in terms of accuracy and sensitivity to uncertainties in density–elasticity relationships and load amplitude as model input variables. The load independent behavior of SR and its direct clinical interpretation may ultimately provide an appropriate and easily understood comparative computational measure to choose between patient specific fracture fixation alternatives [30]. Press-fit conditions which allow implantation without excessive plastic bone deformation and sufficient primary stability to achieve bone in growths was determine. The influence of interference between, bone quality and friction on the micro motion during walking and stair-climbing was investigated. Therefore elastic and plastic finite element (FE) models of the proximal femur were developed and implantation was realized by displacing the prosthesis into the femur while monitoring the contact pressure, plastic bone deformation as well as implantation forces. Subsequently a physiologic gait and stair-climbing cycle was simulated calculating the micro motion at the bone-implant interface [33]. Fully bonded and frictional interfaces were investigated for combinations of three proximal femurs and two implant designs, the proximal short stem and the IPS hip stem prostheses. The Monte Carlo method was used with two performance indicators: the percentage of bone volume that exceeded specified strain limits and the maximum nodal micro motion. The six degrees of freedom of bone-implant relative position, magnitude of the hip contact force, and spatial direction were the random variables. The distal portion of the proximal femurs was completely constrained and some of the main muscle forces acting in the hip were applied. The coefficients of the linear approximation between the random variables and the output were used as the sensitivity values. In all cases, bone- implant position related parameters were the most sensitive parameters [11]. Hypothesize that variability in knee subchondral bone surface geometry will differentiate between patients at risk and those notate risk for developing osteoarthritis (OA) and suggest that statistical shape modeling (SSM) methods form the basis for developing a diagnostic tool for predicting the onset of OA. Using a subset of clinical knee MRI data from the osteoarthritis initiative (OAI) determine firstly utilize SSM to compactly and efficiently describe variability in knee subchondral bone surface geometry and also the efficacy of SSM and rigid body transformations to distinguish between patients who are not expected to develop osteoarthritis and those with clinical risk factors for OA [3]. Measured the stem-bone micro motion and implant cup-bone relative displacements in addition to surface strains at different locations and orientations on the proximal femur and to compare these measurements with those predicted by equivalent FE models. The loading and the support conditions of the experiment were closely replicated in the FE models. A new experimental set-up has been developed, with specially designed fixtures and load application mechanism, which can effectively impose bending and deflection of the tested femurs, almost in any direction [29]. A mathematical model of Monte Carlo was developed to simulate three-dimensional femur bone and femur bone with implant in the femoral canal, taking into account stress distribution and total displacement during horizontal walking. The equilibrium equations are used in the model. Realistic domains are created by using CT scan data. Different cases of static loads and different boundary conditions are used in the simulation. The finite element method is utilized to determine total displacement and Von Misses Stress. The influences of human weight during horizontal walking are investigated [1].

Conclusions

Computational modeling techniques applied to the human femur bone and finite element method is used to response of model under well defined boundary conditions. Though mechanical properties of bone are inhomogeneous and its variation depends on individual, it influences on the total stiffness and stress condition of the bone.

From the above literature survey, it may be concluded that three dimensional models of femur bone were generated by most of the authors, using CT/MRI scan data of either dry or frozen human femur bone for finite element analysis and validated the results by experimental analysis. Real dry bone, frozen bone, synthetic bone or rapid prototype model from CAD data is used for experimental analysis of human femur. Some of the authors carried their research on effect of nailing and Total Hip Replacement (THR) on human femur bone by using finite element method. The finite element model was generated, starting from the CT/MRI dataset, using a procedure that can be summarized in the following steps:

- Extraction of the 3D bone model of femur
- Generation of the finite element mesh
- Assigning of inhomogeneous mechanical properties in FE model before analysis

Different types of material properties can be assigned to three dimensional FE model of femur bone by various ways. Heterogeneous material properties assigned by using empirical relationship in terms of bone mineral density and Young's modulus. The material properties to each elemental mesh were assigned using the density information derived from the CT scan data set. Some of the authors used an average Hounsfield Unit (HU) values and was computed for each element performing a numerical integration of the HU field over the elements. MATLAB programming is also used by some authors for assigning material properties. A linear relationship between the HU numbers and the bone ash density was assumed. Since inclination angle of human femur varies with individuals, different authors used it between 7° to 28° under load conditions for analyzing stresses and deflections. The results conclude from various articles that the density-based inhomogeneous models of femur predicts with a very good accuracy the measured stress field, while the homogeneous material models results were less accurate, although not dramatically. The numerically predicted stresses were highly correlated with the experimental results. Lagrangian and Monte Carlo algorithms were used for mathematical modeling. This result was not directly validated, since no measurements were taken inside the bone, nevertheless it is reasonable to think that the model that better predicts the experimental results on the bone surface, will better predict the stress field inside the bone as well. Also the strategy of material properties mapping has a significant effect on the stress prediction, specifically when the stress field in the bulk of the bone is of interest as, for example, when studying the interaction between bone and a prosthetic implant. During experimental validation a large number of strain gauges were used. However, if the bone is affected by a severe pathology that dramatically changes its mechanical characteristics, the results may be no longer valid from experiments. This would require far more work and such studies are beyond the scope of the present work. Software i.e. solid works, CATIA and Materialise MIMICS were used to create three dimensional model of femur bone, while ANSYS, ABAQUS and HYPERMESH is used for FEA analysis.

In this review paper various methods of finite element analysis of human femur bone is illustrated. The mechanical response of an individual patient's bone and the proximal femur in particular, is of major clinical importance for orthopedists.

Conflict of Interest

The authors have no conflicts of interest. The authors hereby declare that with regard to the submission of this article there are no financial and personal relationships with other people and organizations.

References

[1] Amornsamankul S., Kaorapapong K. and Wiwatanapataphee B. (2010) *International Journal of Mathematics and Computers in Simulation*, 4, 171-178.

[2] Andrea Malandrino, Fulvia Taddei, Enrico Schileo, Mateusz Juszczyk, Luca Cristofolini and Marco Viceconti (2008) *Journal of Biomechanics,* 41(S1), 16th ESB Congress, Oral Presentations.

[3] Bredbenner T.L., Eliason T.D., Potter R.S., Mason R.L., Havill L.M. and Nicolella D.P. (2010) *Journal of Biomechanics*, 43, 1780-1786.

[4] Bryan H. Derrickson and Tortora Gerard J. (2005) *Principles of anatomy and physiology,* New York Wiley.

[5] Cheung G., Zalzal P., Bhandari M., Spelt J.K. and Papini M. (2004) *Medical Engineering and Physics*, 26, 93-108.

[6] Coelho P.G., Fernandes P.R., Rodrigues H.C., Cardoso J.B. and Guedes J.M. (2009) *Journal of Biomechanics*, 42, 830-837.

[7] Cristofolini L., Cappello A., Mc Namara B.P., Viceconti M. (2006) *Medical Engineering and Physics,* 18(6), 502-514,90-98.

[8] Simpson D.J. (2006) *A finite element strategy applied to intramedullary nailng of the proximal femur*, *Ph.D. thesis, Brunel University*.

[9] Bartel D.L., Burstein A.H., Santavicca E.A. and Insall J.N. *J.*

Bone Joint Surg. Am., 64(7), 1026(7).
[10]Dianna D. Cody, Gary J. Gross, Fu J. Hou, Horace J. Spencer, Steven A. Goldstein and David P. Fyhrie (1999) *Journal of Biomechanics,* 32, 1013-1020.
[11]Dopico-Gonza´lez C., Andrew M. New and Browne M. (2010) *Journal of Biomechanics*, 43, 512-520.
[12]Gwun Jang I., Yong Kim I. (2008) *Journal of Biomechanics*, 41 2353-2361.
[13]*http://education.yahoo.com/reference/gray/subjects/subject/59*.
[14]*http://en.wikipedia.org/wiki/Bone*.
[15]*http://en.wikipedia.org/wiki/Femur*.
[16]*http://www.britannica.com/EBchecked/topic/204137/femur*.
[17]Sanz-Herrera J.M., Garcıa-Aznar and Doblare M. (2008) *Computer Methods in Applied Mechanics Engineering,* 197, 3092-3107.
[18]Lewis J.L., Askew M.J. and Jaycox D.P., *J. Bone Joint Surg Am*, 64(1), 1294 (1).
[19]Little J.P., Taddei F., Viceconti M., Murray D.W. and Gill H.S. *Response to physiological loads, Clin Biomech* (Bristol, Avon), 22(4), 4402(4).
[20]Jonkers I., Sauwen N., Lenaerts G., Mulier M., Perre G.V., Jaecques S. (2008) *Journal of Biomechanics*, 41, 3405-3413.
[21]Kasiri S., Taylor D. (2008) *Journal of Biomechanics*, 41 603-609.
[22]Katja Hoehn Marieb, Elaine Nicpon (2007) *Human Anatomy & Physiology, 7th Edition*. *San Francisco: Benjamin Cummings*.
[23]Lengsfeld M., Schmitt J., Alter P., Kaminsky J., Leppek R. (1998) *Journal of Medical Engineering and Physics*, 20 515-522.
[24]Lengsfeld M., Kaminsky J., Merz B., Franke R.P. (1996) *Journal of Medical Engineering Physics*, 18(1), 70-78.
[25]Doblare M. and Garcia J.M. *J. Biomech.*, 34(9), 1157(9).
[26]Fagan M.J. (1997) *Theory and practice*, Longman, Singapore.
[27]McNamara B.P., Cristofolini I., Toni A., Taylor D. (1997) *Journal of Biomechanics*, 30(6), 621-630.
[28]Netter, Frank H. (1987) *Anatomy, physiology, and metabolic disorders*. New Jersey, Summit: Ciba-Geigy Corporation.
[29]Pal B., Gupta S., Andrew M.R. New, Browne M. (2010) *Journal of Biomechanics*, 43, 1923-1930.
[30]Peleg E., Beek M., Joskowicz L., Liebergall M., Mosheiff R., Whyne C. (2010) *Journal of Biomechanics*, 43, 2684-2688.
[31]Pise U.V., Bhatt A.D., Srivastava R.K., Warkedkar R. (2009) *Journal of Biomechanics,* 42, 1981-1988.
[32]Huiskes R. and Chao E.Y. *J. Biomech.*, 16(6), 3856 (6).
[33]Rothstock S., Uhlenbrock A., Bishop N., Morlock M. (2010) *Journal of Biomechanics*, 43, 521-526.
[34]Teoh S.H., Chan W.H. and Thampuran R. (2002) *Journal of Biomechanics,* 35, 323-330.
[35]Schmitt J., Meiforth J., Lengsfeld M. (2001) *Medical Engineering and Physics*, 23 529-539.
[36]Sinha A., Kumar A., Garg A., Bhargava A., Mishra A.K., Sharma B., Mishra B., Thakur S.S., and Kumar V. (2009) *International Journal of Computer Science and System Analysis*, 3 (1), 13-16.
[37]Taddei F., Cristofolini L., Martelli S., Gill H.S., Viceconti M. (2006) *Journal of Biomechanics*, 39, 2457-2467.
[38]Trabelsi N., Yosibash Z. and Milgrom C. (2009) *Journal of Biomechanics*, 42, 234-241.
[39]Tsubota K., Suzuki Y., Yamada T., Hojo M., Makinouchi A., Adachi T. (2009) *Journal of Biomechanics*, 42, 1088-1094.
[40]Vijayakumar V. (2003) *Stress/strain environment in healing human tibial fractures, D.Phil. thesis, University of Oxford*.
[41]Waide V., Cristofolini L., Stolk J., Verdonschot N. and Toni A. *Clin. Biomech.*, 18(6), 5238.
[42]Verhulp E., Rietbergen B.V., Huiskes R. (2006) *Journal of Biomechanics*, 39, 2951-2957.
[43]Viceconti M., Davinelli M, Taddei F., Cappello A. (2004) *Journal of Biomechanics*, 37, 1597-1605.
[44]Weinans H., Sumner D.R., Igloria R., Natarajan R.N. (2000) *Journal of Biomechanics*, 33, 809-817.
[45]Xinghua Z., He G. and Bingzhao G. (2005) *Journal of Biomechanics*, 38, 1612-1620.
[46]Yosibash Z., Trabelsi N. and Milgrom C. (2007) *Journal of Biomechanics*, 40, 3688-3699.
[47]Zannoni C., Viceconti M., Pierotti L., Cappello A. (1998) *Journal of Medical Engineering and Physics*, 20, 653-659.