International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 1 Issue 6, August - 2012

Computational Modeling of Human Femur using CT Data for Finite Element Analysis

Amrita Francis[1]
[1]M.E. Student, Government Engineering College, Jabalpur, M.P. – India

Veerendra Kumar[2]
[2]Professor and Principal, Department of Mechanical Engineering, Government Engineering College, Jabalpur, M.P. – India

Abstract

Three-dimensional finite element modeling is widely used to generate reliable subject-specific FE model using Computed Tomography (CT) data that accurately predicts information about bone morphology and tissue density. CT scan data is widely used to make realistic investigations on the mechanical behavior of bone structures using Finite Element Analysis (FEA). The purpose of this paper is to create 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. Analysis of these models will provide data unavailable at this time to orthopaedic surgeons, engineers and researchers of human orthopaedics.

1. Introduction

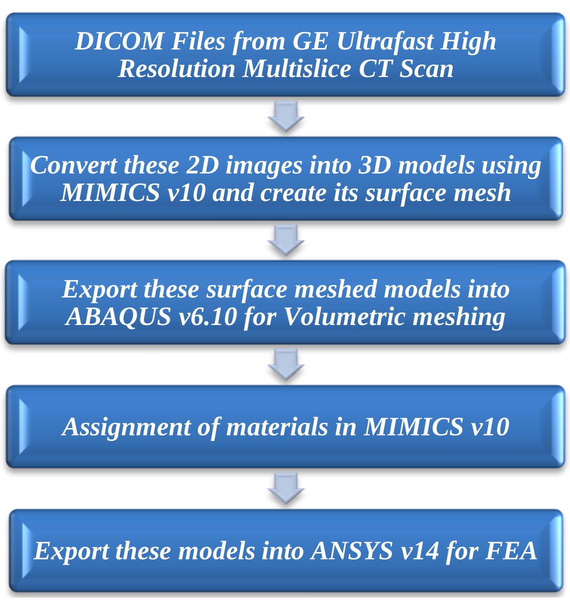
Femur the longest and strongest bone in the skeleton is almost perfectly cylindrical in the greater part of its extent. In the erect posture it is not vertical, being separated above from its fellow by a considerable interval, which corresponds to the breadth of the pelvis, but inclining gradually downward 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 [7]. Being an important structure femur serves two distinct functions: it acts as a supporting structure allowing the weight of the upper body to be transferred from the hip joint to the knee joint and it also acts as a stiff structure about which muscles act to facilitate movement at both the hip and knee joints. 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. Fracture of the shaft of the femur occurs when subjected to extreme force such as in a road traffic accident. Thus, this complete study of human femur is addressed under biomechanics. As biomechanics is the study of motions experienced by living things in response to applied loads. Koch is the first who gave a complete and thorough description of the structure of the femur and demonstrated the relations which exist between the structure and the function as well as between the external and internal architecture of the femur [5]. 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, this structure mainly resists compression and bone elements place or displace themselves in the direction of functional pressure according to Wolff's Law [2].The shape of the femur is asymmetric and curved in all three planes. Hence, a three-dimensional model is required for a quantitative stress analysis [1, 8]. With minor modifications CT scans FE models can be used to generate reliable subject-specific FE models that accurately predicts strains in quasi-axial loading configurations [11, 21, 28, 29]. Thorough understanding and behavior of femur is essential to elucidate the femur fracture and provide better guidance to the artificial femur replacement. Various works has been carried out to investigate the loading mode and stress distribution [30, 14]. For better understanding of femoral loading forces exerted by the soft and hard tissues of the thigh together are considered, a three dimensional model is created taking into account all thigh muscles, body weight, contact forces at the hip, patello-femoral and knee joints [3, 6, 10, 22]. A mathematical model is 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

ISSN: 2278-0181
Vol. 1 Issue 6, August - 2012

walking [26]. Material properties of femur bones are evaluated to facilitate further study of total hip joint and replacement of joint in Indian subjects, as these properties are needed before finite element analysis of indigenized hip joint to study its stability in the bone [19, 27]. The role of ante version in transferring the load from implant to bone and its influence on total hip arthroplasty (THA) is determined. Also loading of the proximal femur during daily activity i.e. walking and stair climbing is determined. Experimental and analytical approaches are used to determine the in-vivo loading of the hip joint. A numerical muscular skeleton model is validated against measured in-vivo hip contact force [6, 16].

2. Materials and Methods

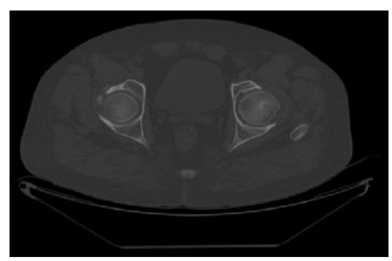
Finite Element Analysis of femur under physiologic conditions is essential for the understanding of failure mechanisms and providing guidance for the design and operation of femur replacement. A three dimensional FE models are created using CT images in Materialise Interactive Medical Image Control System (MIMICS) and various steps involved are described in Fig. 1. For investigation of total deformation, stress distribution and fatigue tools throughout the right proximal human femur under physiological loading conditions FEA is done.



Iĕ

2.1. Image acquisition

The CT data's of total femur of normal individual male patients of 17 yrs, 32 yrs and 40 yrs are collected. This geometrical data of real proximal human femur bone are in the form of Digital Imaging and Communications in Medicine (DICOM) files. The CT scanning of patients are obtained using GE Ultrafast High Resolution Multislice CT Scanner (16 Slice) containing total number of 909, 667 and 1714 images respectively, pixel size of 0.7031 mm, 0.8867 mm and 0.9766 mm respectively, slice thickness of 0.4 mm and resolution of 512 x 512. DICOM file is a standard for handling, storing, printing and transmitting information in medical imaging and contains binary data elements. In MIMICS distinctive CT images are a pixel map of the linear X-ray attenuation coefficient of tissue. The pixel values are scaled so that the linear X-ray attenuation coefficient of air equals -1024 and that of water equals 0. This scale is called the Hounsfield scale after Godfrey Hounsfield, one of the pioneers in computerized tomography. Using this scale, fat is around -110, muscle is around 40, trabecular bone is in the range of 100 to 300 and cortical bone extends above trabecular bone to about 2000. The pixel values are shown graphically by a set of gray levels that vary linearly from black to white [17].



2.2. Image segmentation

ıĕ

Three Dimensional models of right proximal human femur of three individual male human patients are created: Model 1 of 17 yrs old male, Model 2 of 32 yrs old male and Model 3 of 40 yrs old male. Materialise's Interactive Medical Image Control System (MIMICS) is an interactive tool for the visualization and segmentation of CT images as well as MRI images and 3D rendering of objects. Therefore, in the medical field MIMICS is used for diagnostic, operation planning or rehearsal purposes. The Fig. 2 shows the image of normal individual total femur as acquired from the DICOM file images after conversion in MIMICS v10. Bone tissue is then extracted by means of thresholding using default values range from 226 HU to 3071 HU. The extracted bone tissue is put into a mask of volume

1761508.9874 mm^3, 1560018.1976 mm^3 and 1841722.1204mm^3 respectively and number of pixels as 2850426 pixels, 1984073 pixels and 3089900 pixels respectively. These pixels in the marks are modified using various tools successively: edit masks, region growing, and calculation of 3D mask. Edit mask is used to separate the whole total femur from the adjoining hard tissues like pelvis, tibia, etc. The region growing tool provides the capacity to split the segmentation into separate masks with following properties:

Model 1- minimum -238 HU to maximum 1695 HU, number of pixels 143227, mask volume 88511.5585mm^3,

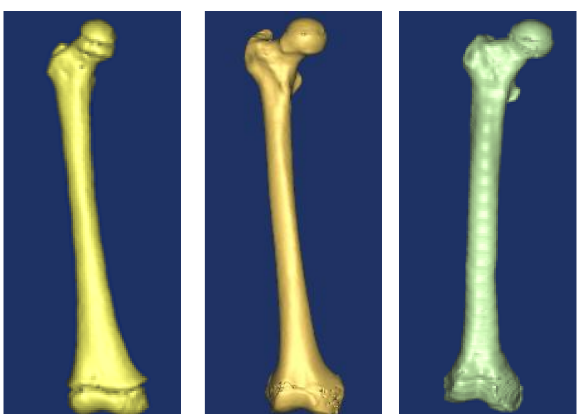
Model 2- minimum -150 HU to maximum 1630 HU, number of pixels 492348, mask volume 387118.7399 mm^3,

Model 3- minimum -118 HU to maximum 1744 HU, number of pixels 445600, mask volume 265598.0377 mm^3, respectively.

Calculate 3D from mask tool converts the 2-Dimensional images into 3-Dimensional models using an interpolation algorithm embedded in MIMICS as shown in Fig. 3, with following properties of different models:

Model 1- mask volume 102228.88 mm^3, surface 26172.09mm^2, triangles 42360 and points 21230,

Model 2- mask volume 471921.39 mm^3, surface 134009.90 mm^2, triangles 140502 and points 69989,



Model 3- mask volume 328208.31mm^3, surface 115704.15 mm^2, triangles 146142, points 72871.

Model 1 Model 2 Model 3

Õë

2.2. Creation of FE model

After creating 3-D models in MIMICS, surface mesh is generated as shown in Fig. 4, for femur bone models for further Finite element analysis. FEA Remeshing is a tool in MIMICS that allows us to create surface mesh of 3-D models. In automatic remeshed operation surface mesh of equilateral triangle is generated. All parameters are optimized here by optimize based method using the ratio of the height of the triangle and the length of its base with quality parameter value above the maximum threshold 0.3. The various other actions performed in FEA remeshing are:

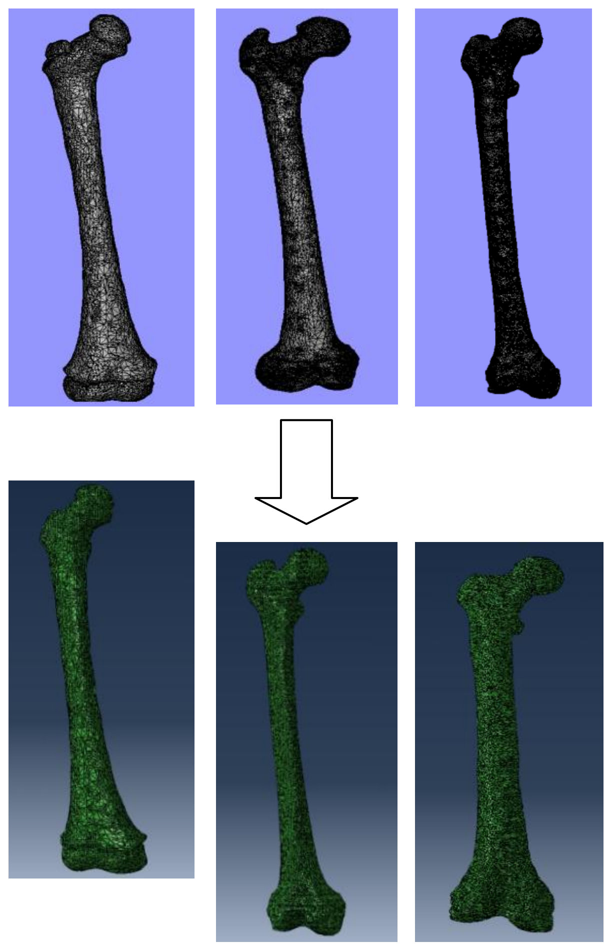
- Triangle reduction of models normally.
- Improving qualities of the triangles.
- Triangle reduction of models quality preserving.
- Removal of extra shells.
- Self intersection test to eliminate intersecting triangles completely.

Properties after remeshing of different models:

Model 1- volume 102094.20 mm^3, surface 26160.67 mm^2, triangles 23816, points 11952,

Model 2- volume 471900.53 mm^3, surface 133994.83 mm^2, triangles 125884, points 62644,

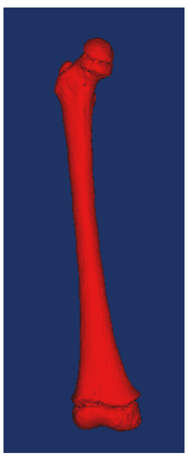
Model 3- volume 328123.40 mm^3, surface 115667.27 mm^2, triangles 128374, points 63953.

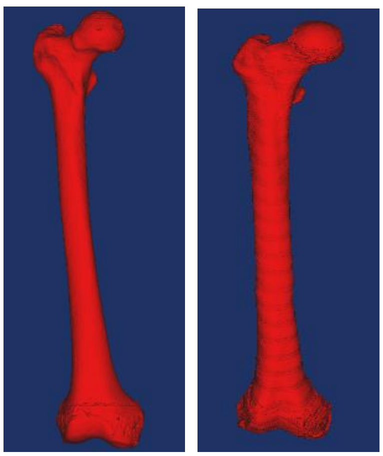


Model 1 Model 2 Model 3

ŏ.

These models are now imported in ABAQUS 6.10 to convert surface mesh into volumetric mesh. In mesh tool choosing edit mesh category and then select convert tri to tetra generates volumetric meshed models. This volumetric meshing converts triangular elements of surface mesh into four node linear tetrahedron elements (C3D4). The three models are meshed with following number of 3-Dimensional tetrahedron elements: 268858, 959083 and 809524, respectively. Fig. 3 shows volumetric meshed models of femur in ABAQUS. From ABAQUS these volumetric meshed models are again imported into MIMICS for material assignment. Fig. 5, shows volumetric meshed models after they are imported in MIMICS.



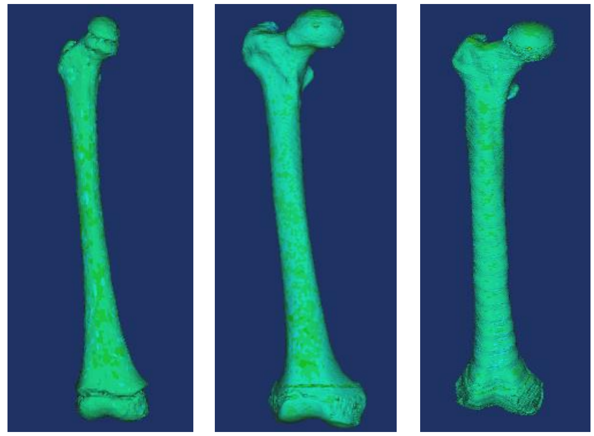


Model 1 Model 2 Model 3

.

2.3. Material Assignment

Femur is a complicated structured material composed of compact bone and cancellous bone. The compact bone is anisotropic material while cancellous bone can be considered as isotropic material. Thus, it is difficult to assign material properties along each direction of bone model. In our study material is assigned to all the models in MIMICS. In MIMICS there are three ways to assign material properties: uniform, look up file and mask. We have considered uniform method for material assignment in MIMICS. In our study ten materials are assigned to each model and gray values are calculated for each material before material assignment. Fig. 6, shows all three models assigned with materials in MIMICS.



Model 1 Model 2 Model 3

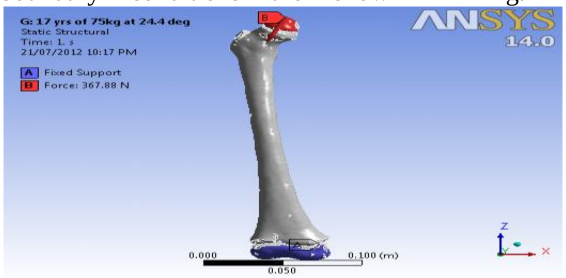
ŏĚ ě

2.4. FE Analysis

The three dimensional Finite Element Models of femur bone with volumetric mesh is imported in ANSYS v14. Since the femur bone models are nonlinear, asymmetric and curved in all three planes, models are first imported in Finite Element Modeler then transfer to static structural module in ANSYS for FEA.

2.4.1 Boundary conditions

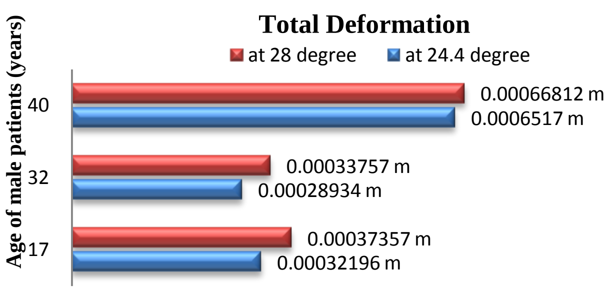
Femur is a thigh bone and shares the whole body weight equally by both the left and right femurs. In our study the actual body weight of 75 Kg (735.75 N), 72 Kg (706.32 N) and 66 Kg (647.46 N) are considered to be equally shared by both femurs according to hip mechanism. Thus, load of 367.875 N, 353.16 N and 323.73 N which is half of whole body weight is applied on right femur head of every model at an angle of 24.4° and 28° [4, 15, 24, 30] and a fixed support is provided at lateral condyle, medial condyle and patellar surface in every model. The boundary conditions are shown in Fig. 7.



ĕ

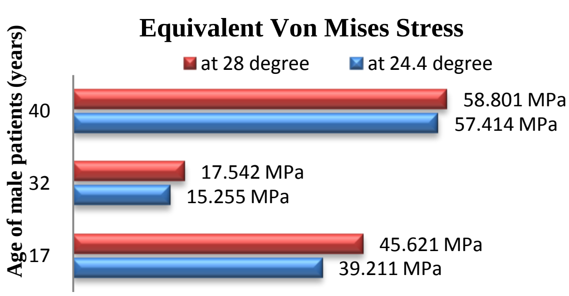
3. Results

The total deformation, equivalent von mises stress and maximum principal stress evaluated in the three dimensional Finite Element Analysis is shown Fig. 8, Fig. 9 and Fig. 10. A constant Fatigue life of 1e6 is obtained throughout the whole femur for all models of 17 yrs, 32 yrs and 40 yrs human male patients. The safety factor evaluated for each model is shown in Fig. 11. This biomechanical study also shows the percentage variation in each model due to increase in curvature or bend in human femur with age which results in the increase in inclination angle of load application as shown in Table 1.



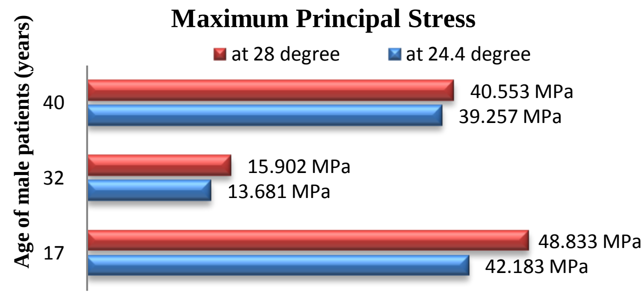
ĕ

1ŏĕŏ° 1 °



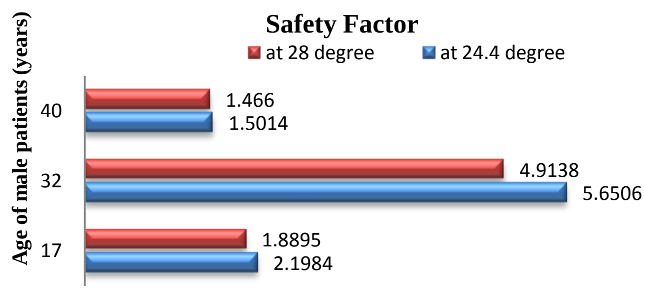
ĕ

1 ° 1ŏĕŏ°



Iĭĕ

1 ° 1ŏĕŏ°



II.

1 ° 1ŏĕŏ°

I.

	Model 1(17 yrs Male)	Model 2 (32 yrs Male)	Model 3 (40 yrs Male)
Total deformation	13.815%	14.287%	2.458%
Equivalent Von Mises Stress	14.051%	13.037%	2.359%
Maximum Principal Stress	13.618%	13.967%	3.196%

4. Discussion and conclusion

Patient-specific bone FE models generated from CT data have become of interest because of their high potential in clinical practice. Although automatic mesh generators may provide good and fast geometrical representation of bones, the determination of their cortical/trabecular sub-domains and associated material properties is still one of the major difficulties in making these FE models reliable enough for clinical applications. Moreover, relatively fewer experimental and computational studies have evaluated the intact whole femur's overall stiffness, strength, cyclic loading and high energy impact loading under various loading regimes, such as axial compression, lateral bending, and torsional loading, which simulate either normal activity of daily living or injury mechanisms. The development of subject-specific finite element (FE) models using computed tomography (CT) data is a powerful tool to noninvasively investigate clinical applications such as fracture risk, prosthesis design and bone remodeling. On applying half of the body weight on the head of the right proximal femur of 17 yrs, 32 yrs and 40 yrs male patients models under study at different inclination angles, following conclusion is investigated:

a) Total deformation increases with increase in the inclination angle.
b) Equivalent Von Mises stress and maximum principal stress increases with increase in the inclination angle.
c) The Bone mineral density is highest in 32 yrs male then 17 yrs male and least in 40 yrs male.
d) The safety factor is highest in 32 yrs male then 17 yrs male and least in 40 yrs male for same body weight.
e) Safety factor is also decreasing with increase in inclination angle of physiologic loading.

Conflict of interest statement

None of the authors have any conflict of interest to declare that could bias the presented work.

Acknowledgment

The authors would like to thank Dr. (Mrs.) Shobha Katheria, Principal Medical Officer, Ordnance Factory Hospital, Itarsi, M.P. India, Dr Rakesh Tirkey, Assistant Professor of Medical College, Jabalpur, M.P, India and Dr. Pushpraj Bhatele, for providing medical imaging data and supporting our work.

References

1. A. Rohlmann, U. Mossner, G. Bergmann and R. Kolbel, Finite element analysis and Experimental Investigation of Stresses in a Femur, Journal of Biomedical Engineering (1982); 4 : 242-246.
2. Aziz Nather, H. J. C. Ong and Zameer Aziz, Bone grafts and bone substitutes – Basic science and clinical applications, World Scientific Publishing Co. Pte. Ltd, Structure of bone; 5.
3. Bitsakos, J. Kerner, I. Fisher and A. A. Amis, The effect of muscle loading on the simulation of bone remodelling in the proximal femur, Journal of Biomechanics (2005); 38 : 133–139.
4. D. Testi, M. Viceconti, F. Baruffaldi and A. Cappello, Risk of fracture in elderly patients: a new predictive index based on bone mineral density and finite element analysis, Computer Methods and Programs in Biomedicine (1999); 60 : 23–33.
5. E.E. Gdoutos, D.D. Raftopoulos and J.D. Baril, A critical review of the biomechanical stress analysis of the human femur, Journal of Biomaterials (1982); 3 : 2-8.
6. Georg N. Duda, Erich Schneider and Edmund Y.S. Chao, Internal forces and moments in the Femur during walking, Journal of Biomechanics (1997); 30 (9) : 933-941.
7. H. G. Hanumantharaju and Dr. H. K. Shivanand, Strength analysis & comparison of SS316L, Ti-6AI-4V & Ti-35Nb-7Zr-5Ta used as orthopaedic implant materials by FEA, 2nd International Conference on Chemical, Biological and Environmental Engineering (ICBEE 2010): 101-105.
8. Hagen Wille, Ernst Rank and Zohar Yosibash, Prediction of the mechanical response of the femur with uncertain elastic properties, Journal of Biomechanics, accepted in Feb 2012.
9. Ilse Jonkers, Nicolas Sauwen, Gerlinde Lenaerts, Michiel Mulier, Georges Vander Perre and Siegfried Jaecques, Relation between subject-specific hip joint loading, stress distribution in the proximal femur and bone mineral density changes after total hip replacement, Journal of Biomechanics (2008); 41: 3405–3413.
10. J. A. Simoes, M. A. Vaz, S. Blatcher and M. Taylor, Influence of head constraint and muscle forces on the strain distribution within the intact femur, Medical Engineering and Physics (2000); 22 : 453–459.
11. L. Grassi, E. Schoileo, F. Taddei, L. Zani, M. Juszczyk, L. Cristofolini and M. Viceconti, Accuracy of finite element predictions in sideways load configurations for the proximal human femur, Journal of Biomechanics (2012); 45 : 394-399.
12. Luca Cristofolini, Giorgia Conti, Mateusz Juszczyk, Sara Cremonini, Serge Van Sint Jan and Marco Viceconti, Structural behaviour and strain distribution of the long bones of the human lower limbs, Journal of Biomechanics(2010) ; 43 : 826–835.
13. M. Doblare and J.M. Garcia, Application of an anisotropic bone-remodeling model based on a

damage-repair theory to the analysis of the proximal femur before and after total hip replacement, Journal of Biomechanics (2001); 34 : 1157–1170.

14. M. E. Taylor, K. E. Tanner, M. A. R. Freeman and A. L. Yettram, Stress and strain distribution within the intact femur : compression or bending?, Medical Engineering Physics, Elsevier Science Ltd (1996); 18(2) :122-131.
15. Marco Viceconti, Luisella Bellingeri, Luca Cristofolini and Aldo Toni, A comparative study on different methods of automatic mesh generation of human femurs, Medical Engineering & Physics (1998); 20 : 1–10.
16. Markus O. Heller, Georg Bergmann, Georg Deuretzbacher, Lutz Claes, Norbert P. Haas and Georg N. Duda, Influence of femoral anteversion on proximal femoral loading: measurement and simulation in four patients, Clinical Biomechanics (2001); 16 : 644-649.
17. Materialise's Interactive Medical Image Control System (MIMICS) 10.01: User Manual, Materialise, http://www.materialise.com/mimics/.
18. Mateusz Maria Juszczyk, Luca Cristofolini and Marco Viceconti, The human proximal femur behaves linearly elastic upto failure under physiological loading conditions, Journal of Biomechanics (2011); 44 : 2259–2266.
19. Mrudula S. Kulkarni and S.R. Sathe, Experimental determination of material properties of cortical cadeveric femur bone, Trends Biomaterial & Artificial Organs (2008); 22(1) : 9-15.
20. Nina S. Sverdlova and Ulrich Witzel, Principles of determination and verification of muscle forces in the human musculoskeletal system: Muscle forces to minimize bending stress, Journal of Biomechanics (2010); 43: 387–396.
21. Nir Trabelsi, Zohar Yosibash and Charles Milgrom, Validation of subject-specific automated p-FE analysis of the proximal femur, Journal of Biomechanics (2009); 42 : 234-241.
22. R. A. Brand, R. D. Crowninshield, C.E. Wittstock, D.R. Pederson, C.R. Clark and F.M. Van Krieken, A model of lower extremity muscular anatomy, Journal of Biomechanics (1982); 104 : 304-310.
23. R. Shahar, L. Banks-Sills and R. Eliasy, Stress and strain distribution in the intact canine femur: finite element analysis, Medical Engineering & Physics (2003); 25 : 387–395.
24. Raji Nareliya and Veerendra Kumar, Biomechanical analysis of human femur bone, International Journal of Engineering Science and Technology (2011); 3(4) : 3090-3094.
25. Ryan Krone and Peter Schuster, An investigation on the importance of material anisotropy in Finite Element Modeling of the human femur, SAE International; paper number 2006-01-0064.
26. Somkid Amornsamankul, Kamonchat Kaorapapong and Benchawan Wiwatanapataphee, Three-dimensional simulation of femur bone and implant in femoral canal using finite element method, International Journal of Mathematics and Computers in Simulation (2010); 4 (4) : 171-178.
27. Travis A. Burgers, Jim Mason, Glen Niebur and Heidi L. Ploeg, Compressive properties of trabecular bone in the distal femur, Journal of Biomechanics (2008); 4 : 1077-1085.
28. Z. Yosibash, N. Trabelsi and C. Milgrom, Reliable simulations of the human proximal femur by high-order finite element analysis validated by experimental observations, Journal of Biomechanics (2007a); 40 : 3688–3699.
29. Z. Yosibash, R. Padan, L. Joscowicz and C. Milgrom, A CT-based high-order finite element analysis of the human proximal femur compared to in-vitro experiments, ASME Journal of Biomechanics Engineering (2007b); 129 (3) : 297–309.
30. ZHAO Jun-hai, MA Shu-fang, WEI Wue-ying, Finite Element Analysis of femur stress under bending moment and compression load, IEEE (2009).