Annals of Radiation Therapy and Oncology

Research Article
Published: 25 Feb, 2019

A Finite Element Model for Predicting the Biomechanical Behavior of the Human Femur Affected by a Bone Metastasis

Pusceddu C[1]*, Marrocu A[2], Ball N[3], Melis L[4] and Fancellu A[5]

[1]Department of Oncologic Radiology, Businco Hospital, Italy

[2]Department of Research and Development, Technological Transfer Sector Sardegna Ricerche, Italy

[3]Department of Oncologic Radiology, Regional Referral Center for Oncologic Diseases, Italy

[4]Department of Oncologic Radiology, Division of Nuclear Medicine, Businco Hospital, Italy

[5]Department of Medical Surgical and Experimental Sciences, University of Sassari, Italy

Abstract

Objective: To develop a biomechanically validated Finite Element Model (FEM) to predict the biomechanical behavior of the human femur in patient affected by a large lytic metastasis at high-risk of fracture.

Materials and Methods: 3D geometric models of the femur, device and tumor have been presented, which integrated the CT data-based anatomical structure. Based on the geometric model, a 3D finite element model of a femur was created. Loads, which simulate the pressure from above were applied to the FEM, while a boundary condition describing the relative femur displacement is imposed on the FEM to account for 3D physiological states. The simulation calculation illustrates the stress and strain distribution and deformation of the femur. The method has two characteristics compared to previous studies: FEM of the femur are based on the data directly derived from medical images CTs; the result of analysis will be more accurate than using the data of geometric parameters.

Results: FEM of the real human femur and surgically altered state were loaded with the same force (in accordance with the specifications defined by ISO 7206). The results of the intact and surgically altered state were compared. As they were close together, the FEM was used to predict: load-sharing within tumorous human femur in compression and the stabilizing potential of the different femur implants and cement in compression with respect to different E moduli.

Conclusion: FEM may be used to predict the biomechanical behavior of the femur. Moreover, the influence of different femur devices may be predicted.

Keywords: Biomedical; Finite element method; Models; Stability; Metastases

OPEN ACCESS

***Correspondence:**
Pusceddu C, Department of Oncologic Radiology, Division of Interventional Radiology, Businco Hospital, Regional Referral Center for Oncologic Disease, Cagliari, 09100, Italy, Tel: 390706095123; Fax: 390706095208; E-mail: clapusceddu@gmail.com

Received Date: *29 Jan 2019*
Accepted Date: *21 Feb 2019*
Published Date: *25 Feb 2019*

Citation:
Pusceddu C, Marrocu A, Ball N, Melis L, Fancellu A. A Finite Element Model for Predicting the Biomechanical Behavior of the Human Femur Affected by a Bone Metastasis. Ann Radiat Ther Oncol. 2019; 2(1): 1018.

Introduction

Percutaneous Osteosynthesis plus Cementoplasty (POPC) is a minimally invasive technique used for patients with impending pathological fracture of the proximal femur [1-4]. Little is known about the exact distribution of forces within the femur affected by a tumor or the influence of cement injection and femur implants on femur biomechanics [5]. However, additional knowledge concerning the distribution of forces within the femur would be helpful for example to develop femur implants and to perform future procedures [6]. The finite element method is a standard engineering technique in general used in the design of airplanes, machinery and bridges [7-9]. Using special software, it allows modeling of even complex structures by splitting the structure into numerous, simple finite elements each of which are easy to characterize and model mathematically. These elements are connected by nodes and describe the geometry of model. Material properties are assigned to the single elements and simulation of loading of the model is performed using a computer. However, the predictions of the finite element can only be trusted, if the model has been validated. This especially applies for application of finite element approaches in various biological systems due to a huge variety between individuals. Thus, it also applies for application of finite element modeling in the field of POPC research. Validation may, for example, be done in that way, that the predictions of the model are compared to the results of a corresponding *in vitro* analysis. Thus,

Table 1: Mechanical properties of titanium.

Material	E [GPa]	Poisson	Yield Strength [MPa]
Titanium GR5	106	0.34	828

a useful way to validate a finite element model of the human femur may be a comparison to the results of a parallel *in vitro* analysis. If the results of the *in vitro* investigation and the predictions of the finite element model are close together, the predictions may be trusted and the model could be a useful tool for further investigations. A finite element model allows us to repeat experiments, to change parameters, thus analyzing the effect and influence of a single component within the construct investigated. Therefore, a model such as presented here may be useful for first predictions on new femur implants. It may provide important clues for the stabilizing potential, could be used for analyzing of stress patterns, just leading to an optimal design of the implant. Changes of the shape of the implant can be modeled quickly and their effects predicted before the implant is constructed. It does, however, not mean that biomechanical *in vitro* approaches should be replaced by such a model, but would fully complement them.

The aim of this study was to develop a biomechanically validated finite element model to predict biomechanical properties of a real human femur affected by tumor in compression.

In detail, the FEM should be used to predict:

1. Load-sharing within healthy human femur in compression.

2. Load-sharing within tumoral human femur in compression.

3. The stabilizing potential of the different femur implants and cement in compression with respect to different E moduli.

Methods and Materials

The method involves the FEM to analyze the biomechanical characteristics of the femur based on the medical images. It is a numerical method for solving problems of engineering and mathematical physics. The femur is an anatomical component, which bears loads derived from human activity. The finite element model is fitting well for the bone system, which has a complex structure. In present, the doctor and medical engineer can also utilize the finite element analysis to analysis biomedical properties of femur. Its basic principle was to take a continuity, which was consisted of infinite particles and had finite freedoms such as an aggregation with finite elements. We can get the stain and stress distribution of the whole structure by researching the relation between the displacement of particle and force for every element. It is a good method for resolving biomechanical characteristic problem of complex structure.

Geometric model

We developed a reconstruction model of the real femur based on the CT data-based anatomical structure of the femur by using specific reconstruction software. A 66 years old man with no history of present and past femur disease was selected as normal subject. Initially femur component data were taken in the axial direction 3D acquisitions with, which got 90 contiguous slices images from a C-arm CT scan. The CT images had a slice thickness of 1.0 mm, and each image size is 512 × 512. Ethical permission was obtained for the study and the subject gave an informed consent for participating. The CT scans were imported into mimics from materialize where semi-automatic edge detection was carried out. Three-dimensional object was created of each bone and meshed using surface elements. The meshing was carried out using Magics, an automated meshing module within Mimics (Materialise).

Table 2: This table lists the material properties of element used to model the various components of the femur and the complete model consisted of 90,000 elements.

	Cortical Bone	Spongious Bone
E_1 [MPa]	6,982.9	2,029.4
E_2 [MPa]	6,982.9	2,029.4
E_3 [MPa]	18,155.0	3,195.3
G_{XY} [GPa]	4.69	4.69
G_{23} [GPa]	5.61	5.61
G_{31} [GPa]	7.68	7.68
N_{12}	0.40	0.40
N_{23}	0.25	0.25
N_{31}	0.25	0.25

At two different times we obtained two groups of 3D data from the CT scans: pre and post-operative model. In relation to the preoperative acquisition data we simulated two different conditions: absence and ideal positioning of an inserted medical device. The finite element models of the real human femur, ideal positioning and surgically altered state were loaded with the same force, and compared with each other. Its content included space coordinates of key points as well as topologic structure on the surface of the model. We translated the data from the VTK file format to that of the macro file format in order to import the data to the finite element method software Ansys. We finally created the geometric model of the femur segment in Ansys 11.0.

The space coordinates of key points in the VTK software corresponded to the key points in Ansys 11.0. It is convenient to transfer data between the geometric model in VTK and the finite element model in Ansys 11.0. The geometric model, which was imported into Ansys 11.0 was an entity model. We divided it into a grid of element by applying the finite element meshing on it to form the FEM.

The finite element model

The first step was to create three orthotropic, 3D, nonlinear finite element models of (Figure 1): the models of the real human femur, the ideal positioning and the surgically altered state. Details of the models developing had been given and were briefly summarized here: the shape of the femur segment was reconstructed from data obtained from CT scans of patients with impending pathological fracture of the proximal femur, as shown in Figure 1. Each femur component was modeled as a 10-node isoparametric material element Solid 187 using homogeneous and orthotropic material properties [8,9]. The tumor was modeled using solid elements to simulate an incompressible behavior with a low young modulus and a poisson ratio close to 0.4999 [10].

Regarding the implanted medical device was modeled a BIOS SMALL system with length of 80 mm, made of Titanium GR5, having the determined mechanical characteristics (Table 1). In order to appropriately model the contacting areas between device and internal femur surfaces, which must describe sliding consistent with reality, interface regions were modeled using contact elements. In order to obtain a greater understanding of the internal behavior of the structure and to highlight the stress state of its most important stressed points (neck of the femur and trans-trochanteric region),

Table 3: The table shows that the insertion of the device and cement produces a small increase in maximum deformation. This is justified by the increase in stiffness obtained with the cement injection into the tumor region.

	Deformation [mm]	DIFFERENCE [%]
Surgically altered state	4.17	---
Ideal positioning	4.2	+ 0.7
Real human femur	4.0	- 4.3

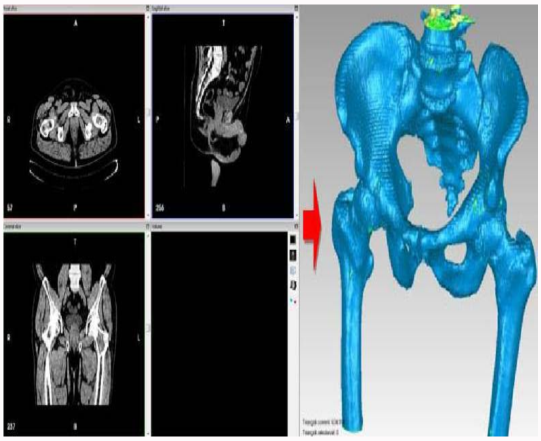

Figure 1: Preparation of the orthotropic, 3D, nonlinear finite element models of the femur real structure. First, the reconstruction from the data obtained from CT scan was performed to recreate the shape of the femur. Than each component of the model was modeled using solid element SOLID 187.

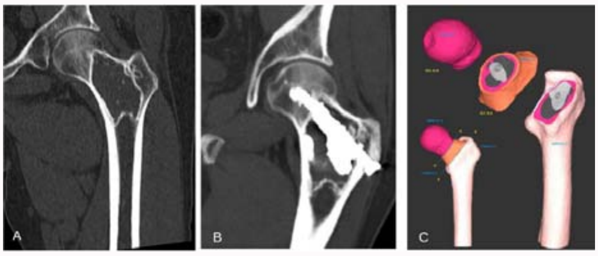

Figure 2: The lateral CT scan shows large lytic metastasis of the neck of the left femur (a). The same scan after treatment shows screw fixation plus cementoplasty (b). In order to obtain a greater understanding of the internal behavior of the structure and to highlight the stress state of its most stressed points (neck of the femur and trans-trochanteric region) the model was divided into three regions, corresponding to: LAYER 1 - head of the femur; LAYER 2 - neck of the femur; LAYER 3 - diaphyseal region of the femur (c).

each model was divided into three regions (Figure 2), corresponding to: LAYER 1 - the head of the femur; LAYER 2 - neck of the femur; LAYER 3 - inter-trochanteric region of the femur.

The material properties used in the study were derived from the literature [11-14]. The behavior of material properties in the model response better reflected those of published experimental femur response. Here, we hypothesize that the strain of femur is a small strain (Table 2).

Boundary conditions

With regard to the validation and accurateness of model analysis, we applied the boundary conditions on the FEM. The computation model is inspired by the specifications defined by ISO 7206, used in the fatigue test of the hip prosthesis. The boundary conditions on the model (Figure 3) use pressure and restrains assigned to surface areas of the model. The inferior surface of femur body was fixed in

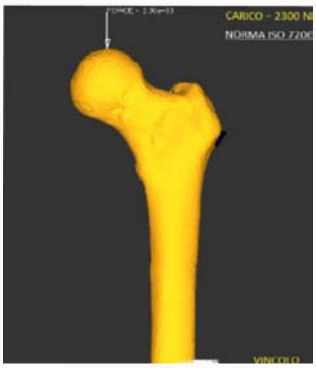

Figure 3: The model was constrained in the distal area, requiring all the nodes at the end of the bone displacements and rotations void in all directions. As regards the application of the load was taken into consideration a force, hypothesized concentrated, which is discharged vertically on the head of the femur, with a value equal to 2300 N, as described by ISO 7206. In particular, since the forces are obviously vary in time and depend on the subject and the type of road in question, it was decided to perform a static analysis refers to the configuration in which it is maximum and the vertical action to apply to the head of the femur only this force.

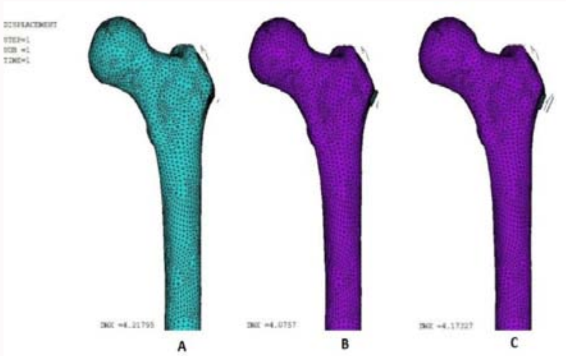

Figure 4: From the picture we can see, for each model, an increase tendency of displacement of the femur with the increase of the distance from the point of distal constraint. The tendency is approximately linear which also illustrates that the femur bone has flexible biomechanical characteristics. The picture shows load displacement for: the real human femur (A), ideal positioning (B) and surgically altered state (C).

all directions. The restraints were used to limit the models movement with six possible values at the node on the surface, three translations and three rotations. The value of freedom was zero.

Load cases

In this paper we will analyze the stress and strain distribution of the femur [15-21]. The evaluation was performed by load-displacement behavior method. We can observe the displacement change of the femur and the strain distribution of the segment under a load of 2,300 N axial compressions, applied to the superior surface of the femur head in the form of a uniformly concentrated load over all femur superior surface nodes. We can observe the stress distribution of the femur segment by applying the load and clue on the high stress concentration region as the most likely areas fracture. From the load cases, we know that the finite element model can be used to predict the change of biomechanical behavior of the femur under pressure.

Results

This study presents the results in three parts. The stress and strain distributions of the femur in the real human femur, ideal positioning

Table 4: Comparative table of results.

	Small Trochanter		Femur Neck			
Sigma Max [MPa]	Node (20,723)	Difference [%]	Node Up (56,633)	Difference [%]	Node Down (60,833)	Difference [%]
Real human femur	100.0	…	19.406	…	49.186	…
Ideal positioning	80.0	-20.0	10.363	-46.6	35.653	-27.5
Surgically altered state	85.0	-15.0	10.716	-44.8	37.236	-24.3

and surgically altered state were obtained from the biomechanical analysis by applying the same axial compression load. The results are presented in the following sections.

Load displacement

In accord with the specifications defined by ISO 7206 the load we applied on the superior surface of the femur head was 2,300 N. The results of load-displacement behavior in axial compression are shown in (Table 3 and Figure 4).

Stress distribution of model

The Figure 5 shows the stress distribution of the femur, in the three models analyzed, when applied 2,300 N loads. It shows that, in all the models, the high stress concentrations are around the neck of the femur and on the trans-trochanteric region due to the way the load applied. That is, they are mainly focusing on the lower region of the small trochanter. These areas show Von Mises stress that ranges gradually from blue, to the Maximum Von Mises stress indicated in red. The stress on the trans-trochanteric region is higher than that around the neck of the femur, which makes it a common place for injures due to loading. The superposition of the effects produced by the mechanical actions provide the most critical conditions (Table 4): in the femoral neck, where there are no maximum values of the three mechanical actions, but there is a minimum resisting section; in the region below the small trochanter (Node 20723), which has the maximum distance from the applied load and, therefore, where you establish the maximum values of the lateral bending. In relation to the neck of the femur, the region most heavily affected involves the outermost fibers, at the top node (Node 56633 UP) and lower (Node DOWN 60833) (Figure 5); regard, however, the small trochanter, the maximum stress is easily identified directly from (Figure 6).

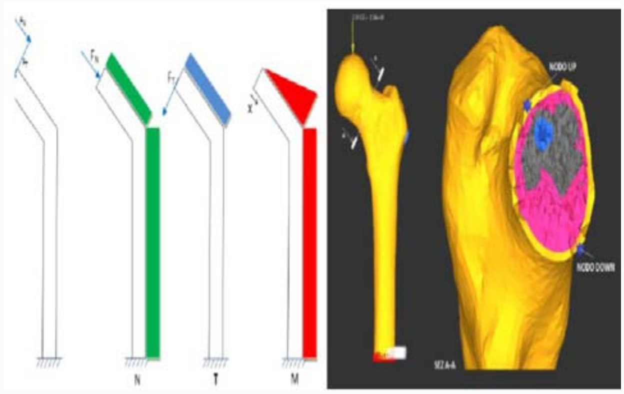

Figure 5: Representation of the superposition of the effects produced by the mechanical actions in the femur. The picture shows that the most critical conditions are: in the femoral neck, where there are no maximum values of the three mechanical actions, but there is a minimum resisting section; in the region below the small trochanter (Node 20723), which has the maximum distance from the applied load and, therefore, where you establish the maximum values of the lateral bending.

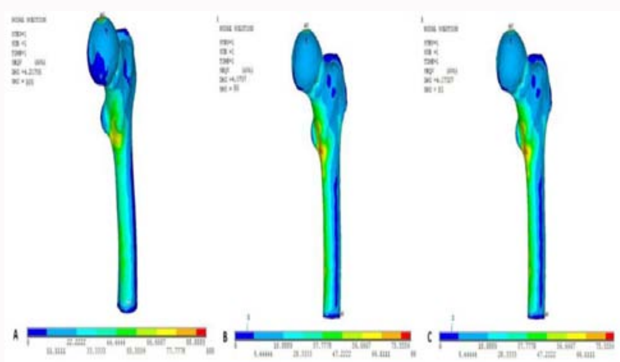

Figure 6: The picture shows Von Mises Stress for: the real human femur (A), ideal positioning (B) and surgically altered state (C). The maximum value of stress obtained for a load of 2300 N is imposed equal to 100.0 MPa, corresponding to the value limit for breaking the cortical bone, and is found just below the lesser trochanter, in conditions of absence of cement and without implanted device; insertion of the device, resulting in the introduction of the cement, reduces by 20% the maximum value of this effort and shows the stress peaks within acceptable limits from the material.

Discussion

The finite element method can be a powerful tool in the field of POPC research. It allows us to repeat experiment, to change parameters, thus analyze the influence of a single component within the construct investigated. It is useful in analyzing stress patterns of femur, also leading to an optimal design of the surgeon. It does, however, not mean that biomechanical *in vitro* approaches should be replaced by such a model. The current finite element model also has limitations, even if its modeling is based on the characteristic of physiological material and the geometric shape of femur. The internal anatomic structure of femur is complicated, and such properties of the small articulation as friction coefficient were not very clear. So all the material parameters adopted for the model were simplified or based on hypothesis on some degree. Any finite element model does only represent a mathematical model and thus is only an approximation to the specimen and even further from real life conditions. It cannot reflect the variability of shape and material properties of the bone inside the individual itself or among the individuals. The interface between two bones only simulates appropriately the condition *in vitro* or *in vivo*. There are lots of differences and uncertain factors induced by the individual diversity during modeling. Based on the above reason, even though a finite element model has some limitations, it simulates the biomechanical characteristics of the femur preferably.

Conclusion

The present study stems from the need to know and analyze numerical methodologies with the states of stress induced by the presence of a tumor region extended in the femur and the potential benefits that can be obtained through the introduction of a screw, able to inject cement within that region. The femur is an important organ for bearing the weight. When load applied on the femur, small distortion appeared and that reflected the flexion properties. It illustrated that the cancellous bone and cortical bone bear the force together. And the high stress is concentrated on the neck and on the trans-trochanteric zone. The high stress is on the inferior region of small trochanter. A 3D nonlinear finite element model of femur was established to simulate the loading state of the component. The study indicates the biomechanical characteristic as follows: the strain of the femur under axial compressive load increased with the performed load; large stress concentrations were found in the trans-trochanteric and neck region, a common place for injures. The results obtained allowed to understand the state of deformation of the bone and the critical areas where occur dangerous stress concentrations. The maximum value of stress obtained for an imposed load of 2,300 N, is equal to 100.0 MPa, corresponding to the breaking limit value for the cortical bone, and is detected just below the small trochanter, in conditions of absence of cement and without implanted device. The insertion of the device, with the consequent introduction of the cement, reduces by 20% the maximum value of this effort and brings the maximum values of stress within acceptable limits from the material. This result can be justified by the fact that the tumor component goes to erode the material of the resistant section just in correspondence of this critical region. A state of stress of this type can lead to a simple trochanteric fracture, with fracture line that extends from the large to the small trochanter (Figure 5).

The last analysis conducted shows that the mechanical point of view, the surgical procedure implementation has allowed to come close to the ideal case initially assumed by the surgeon, with the maximum percentage variations very low, slightly higher than 6% (Table 3). This lighter decrease in results, compared to the ideal case previously illustrated, can be justified by the fact that the quantity of cement inserted has not completely filled the tumor region but has, however, possible to create a "reinforcement" internal able to stiffen the structure and reduce the maximum stress within acceptable limits.

This study enriched some understandings of the biomechanical characteristic under loadings and can help surgeons make better decisions for the treatment with patients with impending pathological fracture of the proximal femur. In the paper, it is an initial model of femur including solid cortical bone, cancellous bone, and the real dimension of the tumor. Our next step is to study more on stress and strain distribution under torsion and shear conditions and to simulate the biomechanical characteristics of femur during an operation. We aim at the operation simulation and surgery navigation by developing and analyzing the finite element model. The finite element model based on medical images can analyze biomechanical characteristics of femur effectively and help optimize individual therapy in the future.

References

1. Raji N, Veerendra K. “Biomechanical analysis of human femur bone”. IJEST. 2011;3(4):3090-4.
2. Perry J. Gait analysis, normal and pathological function. 1st ed. NJ: Slack; 1992. p. 1-47.
3. Paul JP. Comparison of EMG signals from leg muscles with the corresponding force actions calculated from walk path measurements. Human Locomotor Eng. 1971;48A: 6-26.
4. Andriacchi TP, Hurwitz DE. Gait biomechanics and the evolution of total joint replacement. Gait and Posture. 1997;5(3):256-64.
5. Deschamps F, Farouil G, Hakime A, Teriitehau C, Barah A, de Baere T. Percutaneous stabilization of impending pathological fracture of the proximal femur. Cardiovasc Intervent Radiol. 2012;35(6):1428-32.
6. Lengsfeld M, Kaminsky J, Merz B, Franke RP. Sensitivity of femoral strain pattern analyses to resultant and muscle forces at the hip joint. Med Eng Phys. 1996;18(1):70-8.
7. McNamara BP, Cristofolini L, Toni A, Taylor D. Relationship between bone-prosthesis bonding and load transfer in total hip reconstruction. J Biomech. 1997;30(6):621-30.
8. Kenneth HH, Donald LD, Douglas ES, Ted GB. The finite element method for engineers. IV ed. New York: A Wiley-Interscience Publication; 2004.
9. Moaveni S. Finite element analysis theory and applications with ansys. 2nd ed. USA: Prentice Hall; 1999.
10. David SM, Michael EP, Robert JW, Robert D Cook. Concepts and applications of finite element concepts and applications of finite element analysis. 4th ed. Singapore: John Wiley & sons; 2003.
11. Keyak JH, Kaneko TS, Rossi SA, Pejcic MR, Tehranzadeh J, Skinner HB. Predicting the strength of femoral shafts with and without metastatic lesions. Clin Orthop Relat Res. 2005;439:161-70.
12. Hernandez CJ, Keaveny TM. A biomechanical perspective on bone quality. Bone. 2006;39(6):1173-81.
13. Huiskes R, Janssen JD, Slooff TJ. A detailed comparison of experimental and theoretical stress analyses of a human femur. ASME AMD. 1983;45:211-34.
14. Bartel DL, Desormeaux S. Femoral stem performance. In: Weinstein A, Horowitz E, Ruff AW, editors. Retrieval and Analysis of Orthopaedic Implants. Maryland: NBS Special Publication; 1976. p. 51.
15. Franzoso G, Zysset PK. Elastic anisotropy of human cortical bone secondary osteons measured by nanoindentation. J Biomech Eng. 2009;131(2):021001.
16. Frost HM. “Why do bone strength and "mass" in aging adults become unresponsive to vigorous exercise? Insights of the Utah paradigm. J Bone Miner Metab. 1999;17(2):90-7.
17. Doblaré M, García JM. On the modelling bone tissue fracture and healing of the bone tissue. Acta Cient Venez. 2003;54(1):58-75.
18. Cecchini F, Neri P. Analisi strutturale tridimensionale del femore sano. Bologna: ENEA. Dipartimento Reattori Innovativi. 1992;43(6).
19. Neri P. Analisi strutturale e calcolo del carico di rottura nel femore prossimale sottoposto a diverse condizioni di carico. Analisi strutturale del femore con protesi sottoposto ad un carico assiale. Bologna: ENEA, Dipartimento Reattori Innovativi, 29 ottobre 1991.
20. Vichnin HH, Battermann SC. Stress analysis and failure prediction in the proximal femur before and after total hip replacement. J Biomech Eng. 1986;108(1):33-41.
21. Yousif AE, Aziz MY. Biomechanical analysis of the human femur bone during normal walking and standing up. IOSRJEN. 2012;2(8):3-19.