

ORIGINAL PAPER

Numerical simulations of the 3D virtual model of the human hip joint, using finite element method

D. GRECU¹⁾, I. PUCALEV²⁾, M. NEGRU³⁾, D. N. TARNIȚĂ¹⁾,
NINA IONOVICI⁴⁾, R. DIȚĂ¹⁾

¹⁾*Orthopedics and Traumatology Clinic,
Emergency County Hospital, Craiova*

²⁾*Orthopedics and Traumatology Clinic,
Districtual Hospital, Drobeta Turnu-Severin*

³⁾*INAS SA, Craiova*

⁴⁾*Department of Histology,
University of Medicine and Pharmacy of Craiova*

Abstract

In this paper, we present a three-dimensional mathematical model for a normal hip joint. The three-dimensional finite element model has been constructed based on Computer Tomograph scans of the bones. The obtained 3D model is studied using the finite element method, taking into consideration the real structure of the bone and the mechanical characteristics of cortical and spongy. The FE model of hip joint, the material properties used to simulate the behavior of the cortical and trabecular bone (of femur and coxal bone) and the cartilage, as well as the boundary conditions are presented. The distribution map of the axial and global movements on the global model and the distribution map of the axial and von Misses strain in the cartilaginous surface of the femur are presented.

Keywords: hip joint, strains, movements, finite element method.

□ Introduction

Once with the continuous progress of computers technology, graphical and mathematical models have been used more and more in clinical applications. We have used the finite element method in order to virtually represent the hip joint and to study the biodynamic and loads that act on it. We have also tried to link the stress and strain state of the articular surfaces and the forces developed in the muscle structures with the hip pain. And, from another point of view, the study of stress and strain state in the hip joint might prove to be helpful in a preoperative planning (ex. when we have to choose between two types of surgical interventions – hip arthroplasty or osteotomy) [1–3].

A virtual study method of forces that act in the hip joint is the finite element method (FEM), which is based on the Newtonian correlation principles of surfaces that make contact [4–7].

The FEM method was introduced for the first time in 1972 when stress forces that act in human body bones were studied. Since then, the method continued to be used more and more frequently, nowadays being used especially in medical engineering to adapt and to evaluate the endoprostheses [8–15].

FEM is a mathematical method, frequently used in engineering for biomechanics or structural analysis [5, 10, 15–17].

□ Material and Methods

For the theoretical study, FEM was used in the following situation: a normal right hip in one leg standing position, supporting the whole body weight. The hip is subjected to a vertical force of 500 N (equivalent of 60 kg body weight). The main purpose was to determine axial (tensile and compressive) and equivalent strain distribution in the articular cartilage of acetabulum and femoral head. The axial and global displacements of the femur and coxal bone were evaluated.

It was generated a three-dimensional mathematical model for a normal hip joint.

Material

In this study, we have used the normal femur and coxal bone geometry and finite element models generated and thoroughly described in previous papers [16, 17].

We have taken into consideration the real structure of the human bone. We know that the bone is one of the most important natural composite materials. The body of the femur bone is formed by a compact bone tissue cylinder all pierced by a central channel called the medullar channel. The ends of the bone are formed by a thin layer made outside by a compact bone substance, and inside by a spongy mass. The mathematical model of the coxal bone was imported and improved by a delimitation of cortical (variable in thickness) and cancellous bone.

Methods

In order to use FEM it is necessary to follow three stages:

Stage I – Generate the finite element model

In this stage (also known as “Finite Element Model”), with the help of finite element programs available (ANSYS in this study), the geometry of each component is approximated using hexahedron bodies (finite elements), connected by nodes placed in hexahedrons corners. Different attributes were assigned to the hexahedron bodies corresponding to the studied materials, thus obtaining a three-dimensional mathematical model in which we know the three-dimensional position of every node, the geometric characteristics of each finite element (area, volume, mass) and also the stiffness or elasticity of all the components.

Stage II – Applying the loads and known supporting conditions in the real case

This stage is known as „boundary conditions” and consists of known forces and the corresponding application points and known restrained degrees of freedom of the nodes where the structure is sustained. Once these boundary conditions are applied, the mathematical model can be solved.

Stage III – Processing the results

In this stage, the gathered results (type and distribution of displacements, stresses and strains) are visualized in maps with different colors, graphical shapes or lists (tables).

Data processing

The FE model of hip joint, the material properties used to simulate the behavior of the cortical and trabecular bone (of femur and coxal bone) and the cartilage, as well as the boundary conditions applied will be comprehensively presented.

The finite element model

The finite element model of normal hip joint

The finite element model of right acetabulum and right femoral head in which we have introduced the material characteristics such as cortical and trabecular bone and articular cartilage is shown in Figure 1 (a and b). The whole FE model contains 135 724 elements and 156 614 nodes. The acetabulum and the femoral head are positioned in such manner that the cartilage surfaces are in contact as shown in Figure 2. This relative position of the femur and coxal bone was obtained considering the values of the specific joint angles as shown in Figures 3 (a and b) and in Table 1.

The gluteus medius muscle is simulated with a spring element type. The insertion points of the muscle on iliac bone and femur near the tip of greater trochanter are shown in Figure 4. The material properties used are shown in Table 2.

The finite element model of the right coxal bone was appreciated at 48 mm in diameter of the acetabular area (Figure 5). Different material characteristics for cortical and trabecular bone were introduced.

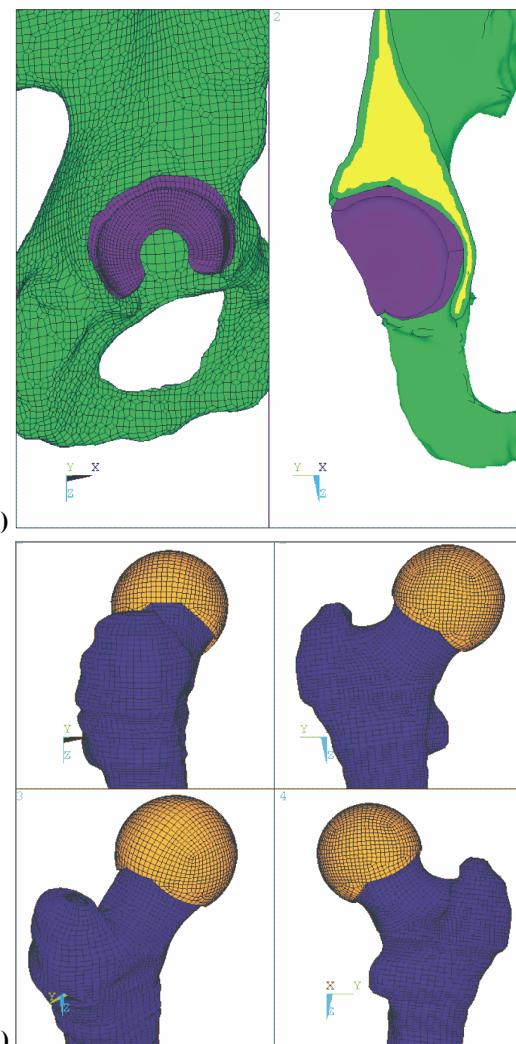


Figure 1 – (a) The finite element model for right acetabulum and articular cartilage – front and profile view (cross-section). **(b)** The finite element model of right femur with the femoral head and articular cartilage – profile, front, oblique and back view.

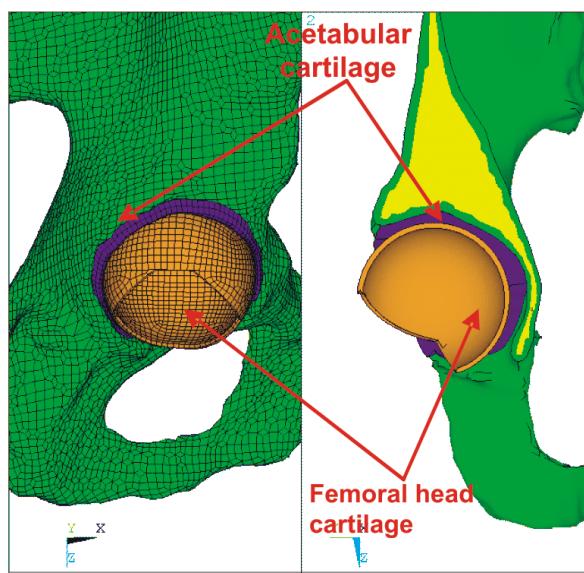


Figure 2 – Finite element model for right hip joint (cartilaginous surfaces of acetabulum and femoral head). Relative view, front and profile (cross-section).

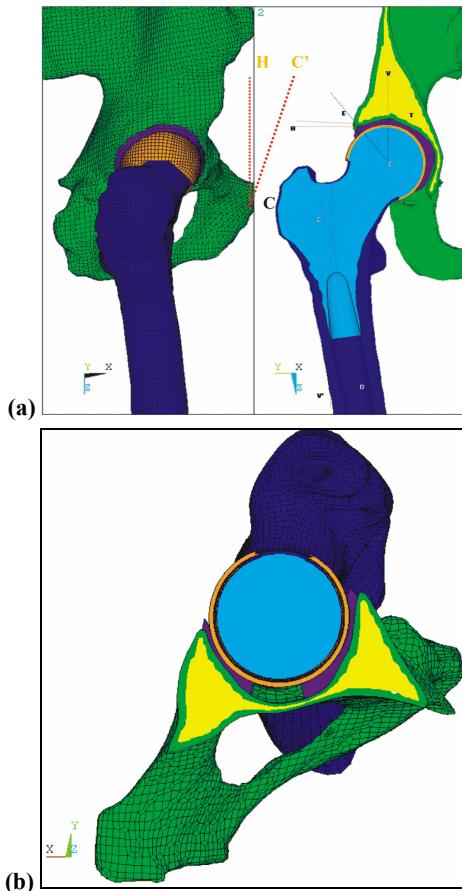


Figure 3 – (a) Finite element model of right hip joint – front and profile view (cross-section). **(b)** Finite element model of right hip joint – upper view (cross-section).

Table 1 – Hip joint specific angles

Angle	Value [degree]	Miscellaneous
CC'D	130	
DC'V'	~8.2	
VCE	~32	
HTE	~2	In standing position, the angle is less than 10 degrees.
HCC'	16	

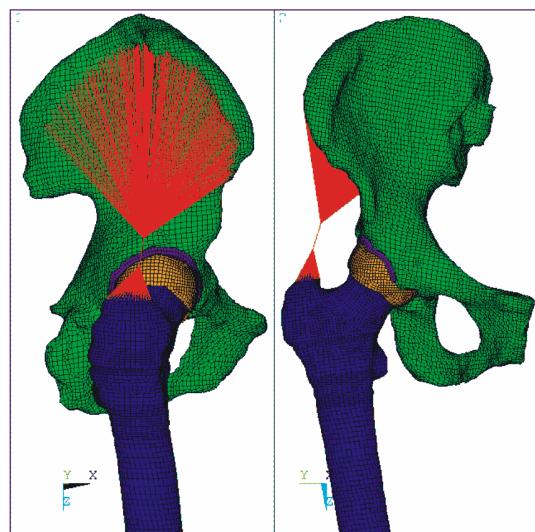


Figure 4 – Finite element model of the right hip joint with simulation of gluteus medius muscle – profile and front view.

Table 2 – Material properties

No.	Component	Material	Young modulus [MPa]	Poisson's ratio
1.		Cortical bone	17000	0.2
2.	Femur	Trabecular bone	1000	0.24
3.		Cartilage	10.5	0.45
4.		Cortical bone	11300	0.3
5.	Coxal bone	Trabecular bone	800	0.2
6.		Cartilage	10.5	0.45

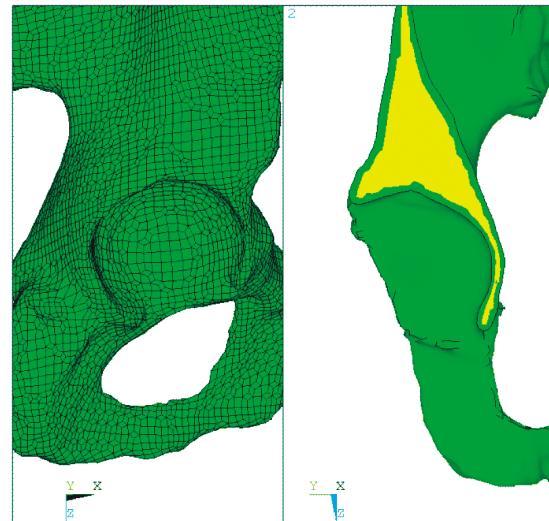


Figure 5 – The finite element model for right hemipelvis – profile and front view.

Boundary conditions

The boundary conditions applied in this study are shown in Figure 6 and consist of:

1. Loads:

- in all the nodes located at zone A (sacro-iliac joint), a total vertical load of $P = 500$ N is applied;
- in gluteus medius muscle a force $FM = 1.6XP = 800$ N is applied [18].

2. Restraints:

- all the nodes located in zone A were restrained for two degrees of freedom, translation along X- and Y-axes ($U_x=U_y=0$);
- all the nodes located at zone B (pubic symphysis) were forced not to move along Y-axis ($U_y=0$);
- the node that simulates the knee joint was considered fixed, the femur can rotate only about X-axis (direction of movement).

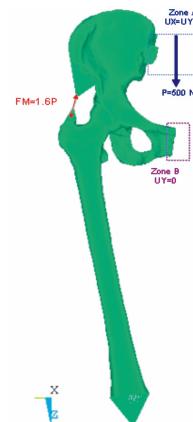


Figure 6 – Finite element model of the right hip joint with simulation of gluteus medius muscle – profile and front view.

□ Results and Discussion

The results gathered, especially the results concerning the femoral head and its cartilaginous surface, will be emphasized. The results were obtained using the von Misses theory. The values are presented in [mm] for axial and global movements. On the value scale, the higher values are indicated by red color and the lower values are indicated by blue color. The values increase from blue to red.

The calculated results are:

- axial and global movements on the global model (Figures 7–10);
- axial and von Misses strain in the cartilaginous surface of the femur (Figures 11–14).

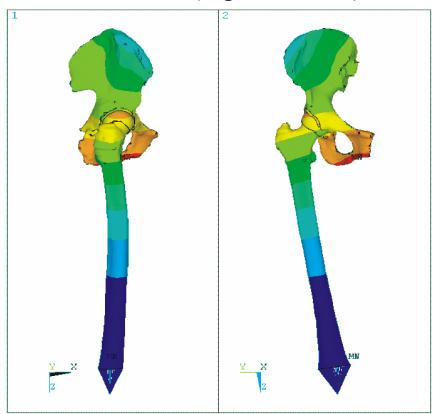


Figure 7 – Axial movements (X-axis) in right hip [mm] – profile and front view.

```
ANSYS 11.0
NODAL SOLUTION
STEP=1
SUB =1
TIME=500
UX (AVG)
RSYS=0
PowerGraphics
EFASET=1
AVRES=Mat
DMX =1.7
SMX =-.3368-03
SMX =1.542
-.3368-03
.096043
.192422
.240611
.336949
.38518
.38519
.529748
.626127
.674317
.770696
.770697
.915264
1.012
1.06
1.156
1.204
1.291
1.349
1.445
1.542
```

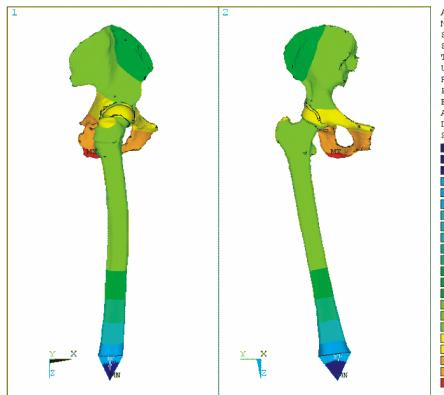


Figure 10 – Global movements in right hip [mm] – profile and front view.

```
ANSYS 11.0
NODAL SOLUTION
STEP=1
SUB =1
TIME=500
EPELY (AVG)
RSYS=0
PowerGraphics
EFASET=1
AVRES=Mat
DMX =1.479
SMX =-.043286
SMX =.025916
-.043286
.034636
.034743
.028148
.028149
.021661
.019498
.015173
.01301
.012495
.00436
.002198
.002128
.00429
.00425
.010778
.015103
.017266
.021591
.025916
```

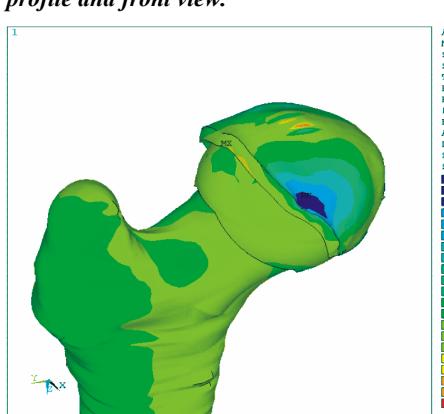


Figure 11 – Axial strains (along X-axis) of cartilaginous surfaces (right femur) – oblique view.

```
ANSYS 11.0
NODAL SOLUTION
STEP=1
SUB =1
TIME=500
EPELY (AVG)
RSYS=0
PowerGraphics
EFASET=1
AVRES=Mat
DMX =1.479
SMX =-.025706
SMX =.025706
-.025706
.018924
.017228
.015977
.012142
.008751
.007055
.003664
.001949
.001422
.004613
.006509
.0099
.01395
.014986
.016682
.020073
.021768
.025159
.028555
```

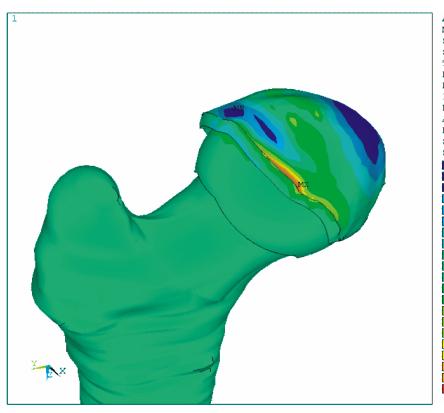


Figure 12 – Axial strains (along Y-axis) of cartilaginous surfaces (right femur) – oblique view.

```
ANSYS 11.0
NODAL SOLUTION
STEP=1
SUB =1
TIME=500
EPELY (AVG)
RSYS=0
PowerGraphics
EFASET=1
AVRES=Mat
DMX =1.479
SMX =-.025706
SMX =.025706
-.025706
.018924
.017228
.015977
.012142
.008751
.007055
.003664
.001949
.001422
.004613
.006509
.0099
.01395
.014986
.016682
.020073
.021768
.025159
.028555
```

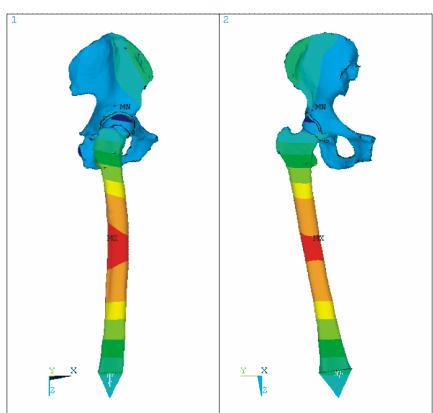


Figure 8 – Axial movements (Y-axis) in right hip joint [mm] – profile and front view.

```
ANSYS 11.0
NODAL SOLUTION
STEP=1
SUB =1
TIME=500
UZ (AVG)
RSYS=0
PowerGraphics
EFASET=1
AVRES=Mat
DMX =1.7
SMX =.29629
SMX =.862831
.0029
.223944
.151397
.115174
.042728
.068505
.102141
.102165
.174611
.210834
.28328
.35277
.39195
.464396
.500619
.573066
.601369
.601735
.717958
.790404
.862851
```

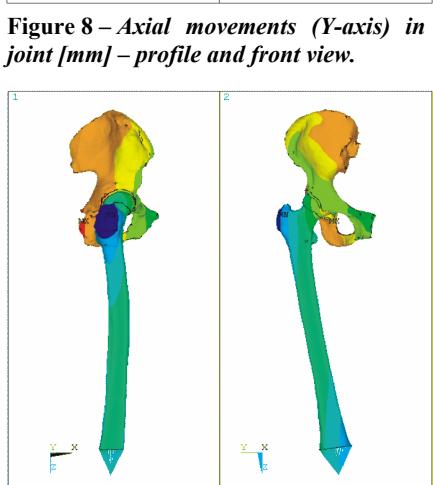


Figure 9 – Axial movements (Z-axis) in right hip joint [mm] – profile and front view.

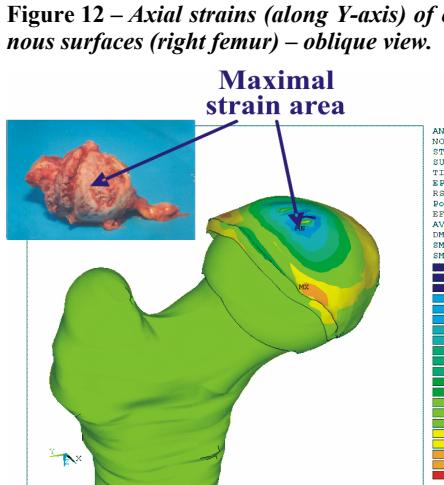


Figure 13 – Axial strains (along Z-axis) of cartilaginous surfaces (right femur) – oblique view.

```
ANSYS 11.0
NODAL SOLUTION
STEP=1
SUB =1
TIME=500
EPELZ (AVG)
RSYS=0
PowerGraphics
EFASET=1
AVRES=Mat
DMX =1.479
SMX =-.023265
SMX =.023265
-.023265
.05912
.049595
.049982
.044556
.041843
.041844
.033705
.026279
.025566
.02411
.014715
.012002
.006577
.00644
.001562
.004275
.004276
.012413
.017639
.023265
```

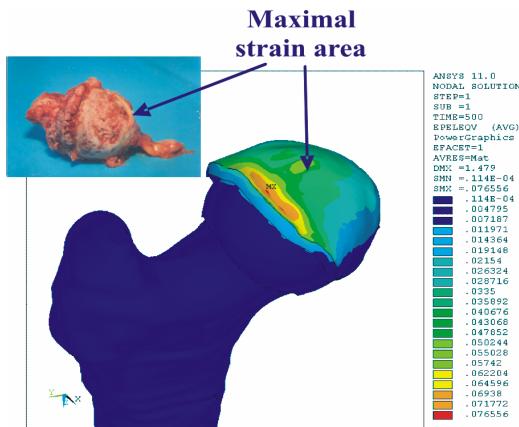


Figure 14 – Von Misses strain in cartilaginous surfaces (right femur) – oblique views.

The maximal strain in the joint, standing on one leg, is at the antero-superior region of the femoral head.

By examining the equivalent and axial strains of the femoral head cartilage it can be noticed that their distribution on Z-axis is almost identical to what was noticed on a real femoral head from a patient who suffered a hip arthroplasty (Figures 13 and 14). The maximum axial strain is -0.064 (6.4%) and corresponds mainly to a compression load along Z-axis (see Figure 13). The maximum equivalent strain is 7.65% and occurs in the inferior region of the femoral head due to a compressive combined load along all three axes.

Conclusions

The distribution of equivalent strain explains the presence of osteophytes on the femoral head because the compression stresses mainly act after Z-axis (main direction). In time, this stressed area wears out, the support capacity of the cartilage in this area (emphasized in Figure 13) diminishes, and osteophytes appear in the inferior region of the femoral head (as shown in Figure 14) as a normal reaction of organism that grows bone in the stressed areas. This also proves that:

- By FEM we show the changes in structure of cartilage and subchondral bone of the femoral head, also shown by a histological macroscopic examination of the femoral head.
- FEM is a non-invasive study method, useful and secure, that can be successfully used in any surgical branch, including orthopedics.
- FEM must be considered as one of the methods useful in a preoperative planning.
- FEM or any other statistical method must be improved in order to more accurate appreciate the real human joint and the lesions that occur at this level and also improved software and hardware might be helpful to mathematically represent and study the modification of the human joint in dynamics.

Corresponding author

Răzvan Diță, MD, Orthopedics and Traumatology Clinic, Emergency County Hospital, 1 Tabaci Street, 200624 Craiova, Romania; Phone +40745–796 875, +40251–510 022, Fax +40251–419 441, e-mail: ban37th@yahoo.com

References

- [1] CHOI K, KUHN JL, CIARELLI MJ, GOLDSTEIN SA, *The elastic moduli of human subchondral, trabecular and cortical bone tissue and the size-dependency of cortical bone modulus*, J Biomech, 1990, 23(11):1103–1113.
- [2] GOLDSTEIN SA, *The mechanical properties of trabecular bone: dependence on anatomic location and function*, J Biomech, 1987, 20(11–12):1055–1061.
- [3] WANG X, WANG T, JIANG F, DUAN Y, *The hip stress level analysis for human routine activities*, Biomed Eng Appl Basis Commun, 2005, 17(3):153–158.
- [4] GULDBERG RE, HOLLISTER SJ, *Finite element solution errors associated with digital image-based mesh generation*, Bioengineering Conference Proceedings of ASME/BED, 1994, 28:147–148.
- [5] HUISKES R, CHAO EY, *A survey of finite element analysis in orthopedic biomechanics: the first decade*, J Biomech, 1983, 16(6):385–409.
- [6] PRATI E, FREDDI A, RANIERI L, TONI A, *Comparative fatigue damage analysis of hip-joint prostheses*, J Biomech, 1982, 15(10):808.
- [7] RYAN TM, SCOTT RS, DUNCAN A, KAPPELMAN J., SHAPIRO L, GRANT S, LEWIS K, STEARMAN R, *Finite element analysis using a 3-D laser scanner*, Am J Phys Anthropol, 1996, 22 Suppl:206.
- [8] BROWN TD, MUTSCHLER TA, FERGUSON AB JR, *A non-linear finite element analysis of some early collapse processes in femoral head osteonecrosis*, J Biomech, 1982, 15(9):705–715.
- [9] JACOBS CR, MANDELL JA, BEAUPRÉ GS, *A comparative study of automatic finite element mesh generation techniques in orthopaedic biomechanics*, Bioengineering Conference Proceedings of ASME/BED, 1993, 24:512–514.
- [10] KOLSTON PJ, *Finite-element modelling: a new tool for the biologist*, Phil Trans R Soc Lond A: Math Phys Eng Sci, 2000, 358(1766):611–631.
- [11] ODGAARD A, KABEL J, VAN RIETBERGEN B, DALSTRA M, HUISKES R, *Fabric and elastic principal directions of cancellous bone are closely related*, J Biomech, 1997, 30(5):487–495.
- [12] SMITH SL, DOWSON D, GOLDSMITH AAJ, *The effect of diametral clearance, motion and loading cycles upon lubrication of metal-on-metal total hip joint replacements*, Proceedings of the Institution of Mechanical Engineers, Part C, J Mech Eng Sci, 2001, 215(1):1–5.
- [13] VAN RIETBERGEN B, ODGAARD A, KABEL J, HUISKES R, *Direct mechanics assessment of elastic symmetries and properties of trabecular bone architecture*, J Biomech, 1996, 29(12):1653–1657.
- [14] VAN RIETBERGEN B, WEINANS H, HUISKES R, ODGAARD A, *A new method to determine trabecular bone elastic properties and loading using micromechanical finite-element models*, J Biomech, 1995, 28(1):69–81.
- [15] ZIENKIEWICZ OC, *The finite element method*, McGraw–Hill, London, 1997.
- [16] GRECU D, *Preoperative planning and postoperative prognosis in total hip arthroplasty*, PhD thesis, University of Medicine and Pharmacy of Craiova, Romania, 2000.
- [17] KOVACS A., *Morphological and clinical considerations upon aseptic loosening of the total hip arthroplasty*, PhD thesis, University of Medicine and Pharmacy of Targu-Mures, Romania, 2004.
- [18] HOBBIE RK, *Intermediate physics for medicine and biology*, Wiley, New York, 1978, 8–13.