U.P.B. Sci. Bull., Series D, Vol. 73, Iss. 3, 2011 ISSN 1454-2358

A COMPUTATIONAL METHOD FOR STRESS ANALYSIS IN HIP RESURFACING

Tudor ATASIEI[1], Lucian Gheorghe GRUIONU[2]

Distribuția tensiunilor mecanice ce iau naştere în matricea osoasă după artroplastia de resurfacing a soldului este evaluata cu ajutorul metodei elementelor finite. Modelul computational al segmentului osos implicat a fost construit pe baza imaginilor furnizate de tomografia computerizata post operatorie "in vivo". Determinarile tensiunilor Von Mises ce apar în matricea osoasa, în cupa iliaca şi în componenta femurală au evidentiat valori de pana la 127 MPa. Simularea efectuată arată ca design-ul implantului şi structura osoasă sunt parametri importanti ai transferului solicitarilor mecanice si în consecință ai integrării osoase şi succesului protezării.

Finite element analysis is employed to estimate bone stress distribution in hip joint after hip resurfacing treatment for osteoarthritis. The numerical model of bone segment was elaborated on the basis of "in vivo" post operative multi sliced computed tomography images. Von Mises stress was evaluated on bone matrix, on the femoral and iliac cups revealing values up to 127 MPa. The simulation shows that implant design and tissue quality are important parameters for the transfer of loads within assemblage, and in conclusion for bone ingrowth and success of prosthetic stability.

Key words: Finite element analysis, hip resurfacing, Von Mises stress

1. Introduction

Hip resurfacing (HR) represents a modern surgical treatment for young and middle-aged active patients with hip osteoarthritis. In the classical total hip replacement, the entire proximal head and neck of the femur is removed. For the HR, the resection is limited to the diseased bone near the articulating surface of the femoral head and the joint is replaced by a metal-on-metal articulation. Its main advantage consists of preserving the femoral bone stock. Thus, subsequent revisions would face primary total hip replacement conditions for the classical femoral component.

Disadvantages of this method are the demanding surgery technique (longer learning curve, risk of malpositioning of the prosthetic components) and the

[1] PhD Student, University of Medicine and Pharmacy "Carol Davila", Bucharest, Romania
[2] Faculty of Technological Systems Engineering and Management, University of Craiova, Romania

difficulty of post-operative investigation of the area inside the femoral component.

In order to investigate how the stress and strain changes after resurfacing surgery in comparison with those occuring in the pre-operative femur, several studies developed computational models. Most of them are performed using the finite element analysis (FEA) which allows the creation of a numerical model of the bone from computed tomography (CT) images of that bone. The reported results are based on CT scans of artificial bones [1,2] or natural bone CT images aquired before implant [3] but none of them used the post operative model.

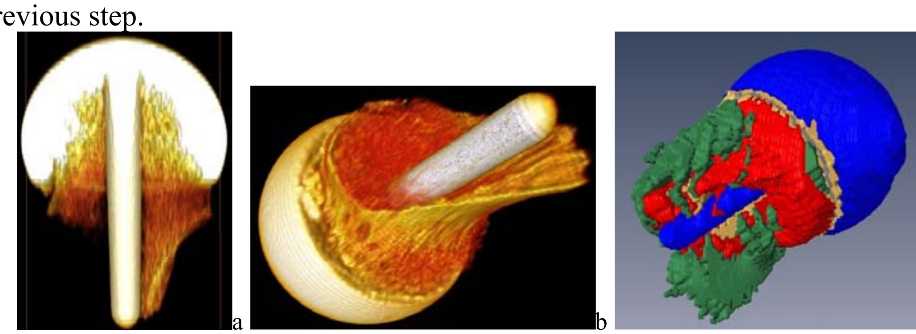
In this article we propose a computational method of assesing the bone matrix inside the femoral component using finite element analysis based on post-operative CT images and joint kinematics of the patient.

2. Methods

Geometrical and mechanical properties of the bone derived from quantitative computer tomography images and 3D CAD models were developed. The images imported on parallel planes have been used to model the exterior surfaces of the joint with resurfacing arthroplasty using an advanced engineering software, Pro/Engineer , as follows:

• on every image, interest zones like bone, implant and cement have been identified and their precise contour shape established (Fig.1, a,b,c),

• spline curves outlining and solid bodies modeling by surfaces obtained at previous step.



a b c

Fig. 1. Tomography pictures of the femoral head with resurfacing arthroplasty (a, b) and material modulus differentiation based on CT scan grayscale value (c)

The resulted assembly contains the following solid bodies: femoral head, femoral head resurfacing cup, iliac bone and its metallic cup which all represent the computational joint arthroplasty morphology and cement used for implant fixation (Fig. 1).

Once the three-dimensional model of the hip was constructed, several anatomical landmarks [4] were located on the joint. Using these landmarks, an individual reference frame of the hip model S_0 was created based on standards of International Society of Biomechanics. The same category of landmarks was identified on patient's hip that was video recorded using two high speed video cameras. This analysis of the video data was performed using SIMI Motion software (Reality Motion Systems GmbH) for every landmark on the physical model and a new reference frame was produced. This non-invasive method is based on recording successive position of several markers placed on the patient's body and limbs that execute normal activities as walking or stair-climbing and post-processing of data for markers trajectories, speed and acceleration computation.

The displacement and rotation vectors for the relocation of the computational model in the global reference frame of the physical model, S_V, were obtained using a numerical optimization procedure [5]. Before optimization, a scale factor was considered for a preliminary resizing of the model. Then, a better fit of landmarks coordinates was obtained according to:

$$s = \sum_i w_i d_i^O \Big/ \sum_j w_j d_j^V \tag{1}$$

where d_i^0 and d_j^V are distances from anatomical landmarks to their reference frame origins for computational and physical model, and w are weighting factors selected on the basis of the distance to the articular surfaces.

The coordinates of markers of the scaled model were used in an error minimization function:

$$\min \Theta = \sum_i w_i \left(\sum_{j=1}^{3} \left(x_{i,j}^O - x_{i,j}^V \right)^2 \right)^{1/2} \tag{2}$$

where $x_{i,j}^O = f(\theta, \gamma)$ are the landmark coordinates of the hip 3D model in S_V, function of displacement γ and rotation θ vector components between S_O and S_V frames, and $x_{i,j}^V$ are the landmarks coordinates on patient.

The resulted motion data can be used in any finite element models for a dynamic analysis.

A precise determination of the density and Young's moduli of the bone is extremely important for the simulation. The three-dimensional model of the joint was meshed with finite elements using Ansys software and the densities of individual elements were produced on the basis of the average greyscale value in the corresponding part of the cross-sectional CT images (Fig. 2, a,b,c,d). Cortical bone density was associated with the highest greyscale value and bone density of 1 g/cm^3 with the lowest greyscale. A linear relationship between the greyscale of the CT scans and apparent density of the bone was assumed. The density was limited by the minimum and maximum densities 1 and 1.95 g/cm^3, respectively.

Young's moduli of the elements representing the bone were determined from their apparent densities using cubic relationship proposed by Carter and Hayes [6], $E=c\rho^3$, where $c=3790$ MPa g^{-3} cm^9. Poisson's ratio of bone was set to 0.35 and the material was assumed to be linearly elastic and isotropic.

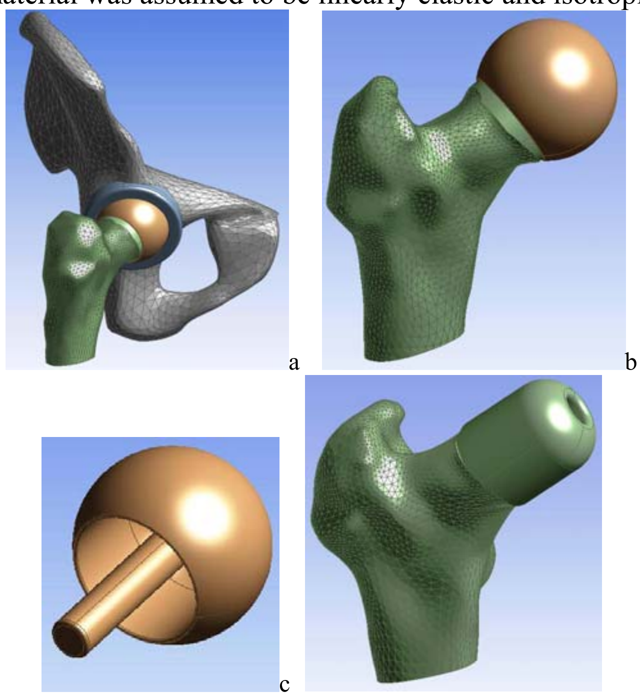
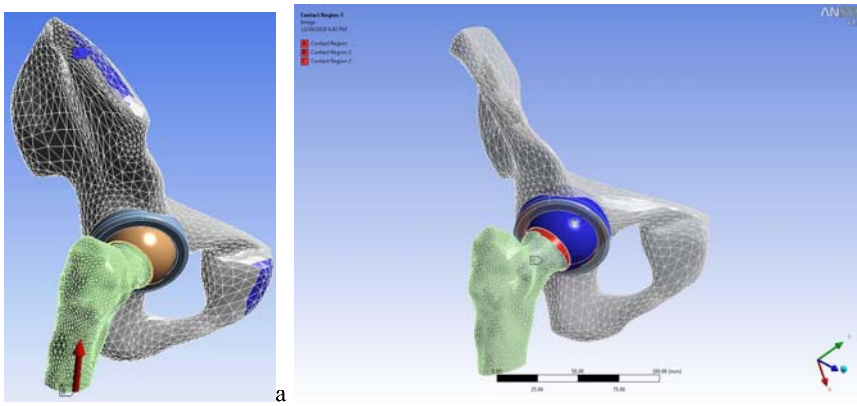


Fig. 2. Finite elements model of the hip joint with resurfacing arthroplasty.

The model (Fig. 3, a) was loaded with the above computed kinematics data for normal walking of the patient one year after resurfacing arthroplasty. The loading consisted of the time functions of femur displacement and rotations for one cycle of walking; that will determine its contact with iliac bone cup. The iliac model was constrained as in anatomic position within the body. Several contact conditions (Fig. 3, b) were imposed to the model as bonding between iliac cup and the bone, and resurfacing cup and the femur. Low friction coefficient (less than 0.07) was imposed to the contact condition between femoral cup and iliac cup.

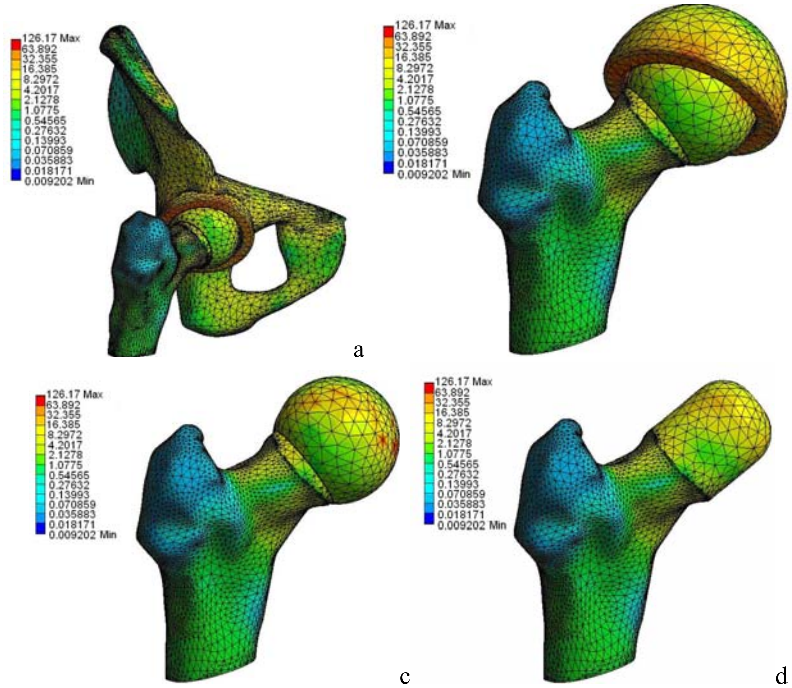


a b

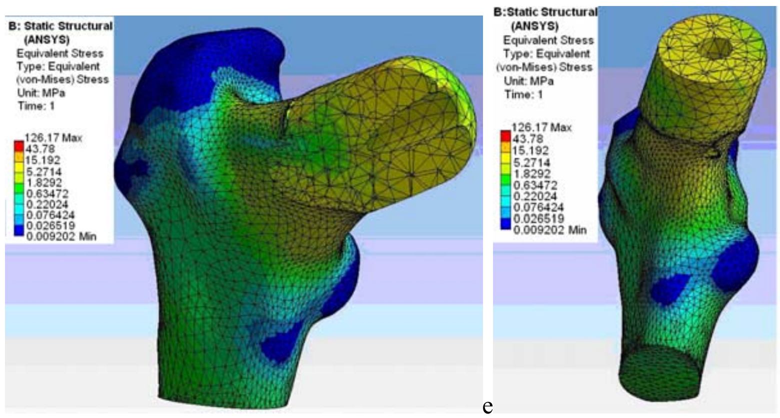
Fig. 3. Model loading and restrain (a) and contact conditions between assembly elements (b).

3. Results

The finite element models for bones and implants were developed from the CAD models using the software packages Ansys to obtain stress pattern (fig. 4) of the bone contact surfaces and implant. During dynamic analysis, the iliac bone was assumed fixed whereas the femoral implant was allowed to move according to computed kinematics data for normal walking. Maximal values of 127 MPa of the stress are observed on the contact surfaces of the femoral and iliac cups.



a b c d



e f

Fig. 4. Results (von Mises stress, MPa) for entire arthroplasty mode (a)l, resurfacing-iliac cups (b), resurfacing cup(c) and femoral bone(d)and virtual section bone sections (e,f) .

4. Discussion

The simulation shows that implant design and tissue quality are important parameters for the transfer of loads within assemblage, and in conclusion for bone ingrowth and success of prosthetic stability. The described procedure including motion acquisition data and finite element analysis, proved to be a valuable tool for arthroplasty investigation. A relatively short area distribution of the stress on the resurfacing cup as presented in the analysis results (Fig. 4, c,d) will conduct in our opinion at a premature iliac cup destruction and it is the result of an insufficient patient joint biomechanics reconstruction after arthroplasty due to implant elements malposition or inter-elements movement after surgery.

R E F E R E N C E S

[1] *A.Heijink, M.E.Zobitz, R.Nuyts, B.F.Morrey, K.N.An*, "Prosthesis Design And Stress Profile After Hip Resurfacing: A Finite Element Analysis", Journal of Orthopaedic Surgery, 2008; **16**(3):326-32

[2] *T.A. Brown, L.Kohan, B.Ben-Nissan* "Assessment By Finite Element Analyis Of The Impact Of Osteoporosis And Osteoarthritis On Hip Resurfacing", 5th ACAM 12-17 dec 2007.

[3] *B.A.Behrens, I.Nolte, P.Wefstaedt, C.Stukenborg-Colsman, A.Bouguecha*, "Finite Element Analysis Of Bone Remodeling After Hip Resurfacing Arthroplasty", 4th IFMBE Proceedings, 2009, vol **22**, part.17, 2288-2291

[4] *VAKHUM Project*, Virtual Animation of the Kinematics of the Human for Industrial, Educational and Research Purposes: Lower Limb Anatomical Landmark Definition and Identification within Different Experimental Contexts. 2002, ISTP.

[5] *C. Bratianu, L.G. Gruionu.* "Computational Simulation Of A Total Knee Prosthesis Mechanical Behavior". Key Engineering Materials, vol. 306-308, pag. 1265-1270, Part 1-2, 2006 ISSN: 1013-9826,

[6] *D.R. Carter, and W.C. Hayes*, "The Compressive Behavior Of Bone As A Two-Phase Porous Structure", J Bone Joint Surg Am, 1977. **59**(7): p. 954-62.