ANNALS of the ORADEA UNIVERSITY.
Fascicle of Management and Technological Engineering, Volume IX (XIX), 2010, NR3

BIOMECHANICAL BEHAVIOR ANALYSIS OF A COXOFEMORAL SHORT STEM IMPLANT

TOTH-TASCAU Mirela*, KREPELKA Mircea*
* Politehnica University of Timişoara, Faculty of Mechanical Engineering
mirela@cmpicsu.upt.ro, mirceakrepelka@gmail.com

Keywords: prosthesis, hip implant, short stem, numerical analysis

Abstract: This paper represents a part of an extensive study that focused on the comparative analysis of three types of custom hip prostheses. The presented study consists in two individual numerical analyses: the natural femur and femur with short-stem implant. The biomechanical behavior analysis of the custom prosthesis was performed using ANSYS Workbench V9.0 program, *Simulation* module. The numerical analysis results showed the stress and deformations of the studied structures, from which we can choose the optimal implant model.

1. INTRODUCTION

A Total Hip Prosthesis (THP) is an artificial hip joint that replaces the patient hip joint and is composed of two components: the femoral (thighbone) component and the cup component that fits into the hip bone.

Each component of the hip prosthesis is designed and manufactured in various shapes and sizes to accommodate various body sizes and types. In some designs, the stem and ball are one piece; other designs are modular, allowing for additional customization in fit.

Thus, there are many models of total hip prostheses on the market and new models appear steadily allowing improvements in long term functionality of the prosthesis [1]. There are two main types of THP: mono-block construction and modular construction, each of them having its own advantages and disadvantages.

Modern Total Hip Arthroplasty (THA) systems are modular. This means that the femoral stem, head, acetabular shell and liner are separate pieces. This modularity allows for greater flexibility in customizing of prosthesis size and fit. The acetabular part is usually a polyethylene liner with or without metal backing. The femoral part is composed of a metal stem (chromium cobalt, titanium or titanium alloy) and a femoral head of metal or ceramic. Stem-fixation is also either with cement or cementless with porous coating for bone in-growth [5].

To solve the problem of a geometric mismatch between the anatomic shape of the femoral canal and conventional stems, and to achieve the best possible fit between vital bone tissue and stem surface, personalized hip implants were developed, using CT scanning, 3D bone reconstruction and CAD modeling.

2. DESIGN ASPECTS

The design aspects (dimensions, radial clearance between the head and the metal socket must be kept as small as possible, surface roughness, etc.) and materials used to manufacture orthopedic implants are of great importance in joint replacements.

The paper proposes a model of a short hip stem implant based on a 3D joint reconstruction of patient images, accomplished with CT scans, Mimics and Magics software.

The design of a short stem implant was started by importing the scanned femur in Solid Edge program and choosing the implant size and position. Being a short implant, it is important that it fills the bone marrow canal of the femur as effectively as possible. The direct sketching on the femur helps to obtain the implant shape and the optimum angle for a normal geometry of the implant, eliminating the risk of dislocations, micro movements, inequality of limb.

A series of angled grooves were created on the side surfaces enabling us to further increase the available surface area and bone in-growth minimizing implant migration.
The acetabular cup consists of two components: a metallic shell and an Ultra High Molecular Weight Polyethylene liner fixed into the shell which acts as a bearing surface due to its low friction coefficient and also absorbs shocks.
The shell fits into the hip joint so there before it needs to copy the shape of the acetabular socket. The fixation is accomplished by using a porous low density Titanium alloy along a series of horizontal grooves on its exterior surface (figure 1).

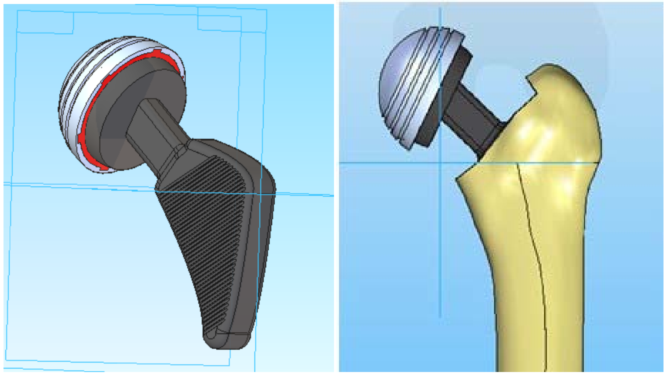


Figure 1. Assembled prosthesis and its position in the femoral bone

The choice of materials depends on many factors, such as: functionality of the implant, the type of interaction with the host organism and the life expectancy of the implant. Thereby, from a large range of biomaterials the chosen ones are the materials with mechanical properties closest to those of the tissue they will interact with.
For the parts that will replace bone tissue (stem and shell), Ti-6Al-4V alloy best matches the condition of mechanical compatibility relative to other metallic materials (figure 2). Metal areas that are subject to friction and have no contact with the bone will be made by polished alloy to gain a superior smoothness to reduce friction. The recommended material for acetabular shells is ultra high molecular weight polyethylene (Ultra High Molecular Weight Polyethylene). It is generally accepted that this polymer is best for use in combination with other biocompatible materials (CoCr alloy, Ti or ceramic), in manufacturing of prosthetic joints. It is a very tough material with high impact resistance, has a low coefficient of friction, self-lubricating and highly resistant to abrasion.

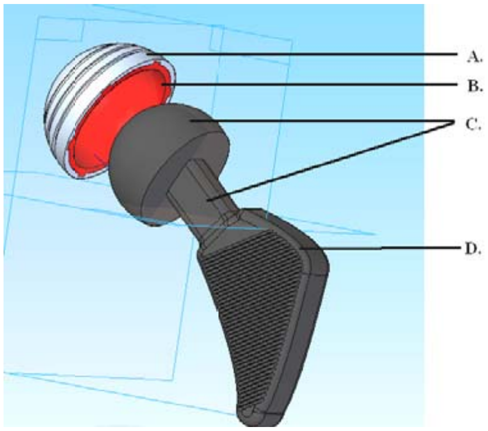


Figure 2. Materials used for hip prosthesis : A - porous Ti-6Al-4V, B - UHMW Polyethylene, C -polished Ti-6Al-4V, D - Ti-6Al-4V

3. NUMERICAL ANALYSIS OF THE FEMUR AND IMPLANTED FEMUR

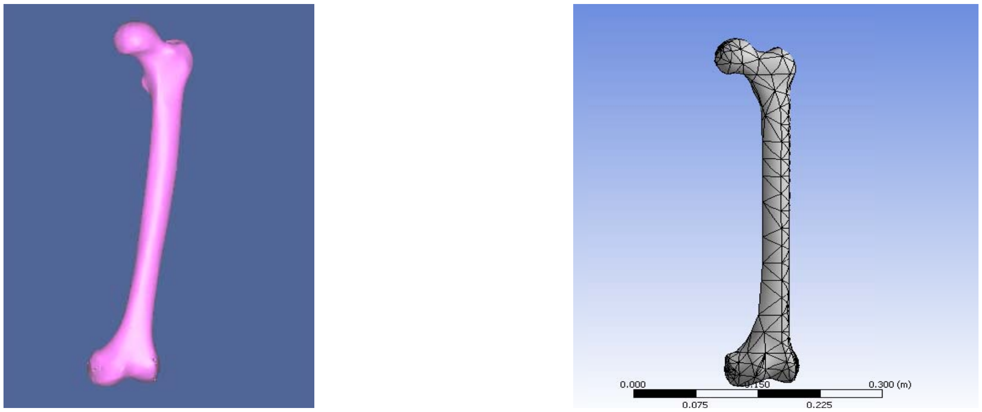
The numerical analysis consists of two separate analyses of components of interest: the natural femur and femur with short stem implant. The results of these tests were then analyzed and compared in order to choose the implant with the best properties.

3.1. Numerical Analysis of the femur

The femur model was obtained based on a 3D joint reconstruction of patient images, accomplished with CT scans and Mimics and Magics software (figure 3.a).

To perform the Finite Element Analysis, the femur is meshed (figure 3.b). The real structure is divided into finite elements, specifying each element position through its nodes connections.

In this stage the topology model is defined on which the calculation is done. Meshing is followed by the material choice, establishing of the fixed points of the piece (figure 4), force positioning and determining of their values.

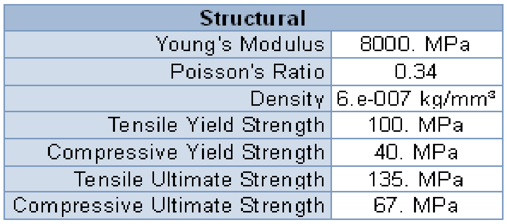


a. Femur model *b. Femur meshing*

Figure 3. Femur model and meshing

For the femur we have chosen the properties of the cortical bone (table1).

Table 1. Mechanical properties of the cortical bone



Structural	
Young's Modulus	8000. MPa
Poisson's Ratio	0.34
Density	6.e-007 kg/mm³
Tensile Yield Strength	100. MPa
Compressive Yield Strength	40. MPa
Tensile Ultimate Strength	135. MPa
Compressive Ultimate Strength	67. MPa

The load environment for bones is complex with forces resulting from joint contact, muscles, tendons, and soft tissues, and the load magnitudes and directions vary as the person moves [2], [3], [4]. Numerical analysis with ANSYS considers that the muscle forces and joint reactions are applied as extensions (directional pressures) [2].

The load values and directions correspond to a static load case representing the stance phase of gait for a person having 80 kg weight.

Thus we have chosen the distal end of the femur as a fixed support and considered two forces (figure 4, table 2): F1 applied on the tip of the femoral head representing the joint reaction, and F2 representing the abductor force [2]. Because fixing the distal end of the femur and lacking effort distribution on the entire leg the test results only show the real distributions and strains on the assembly, values not being precise.

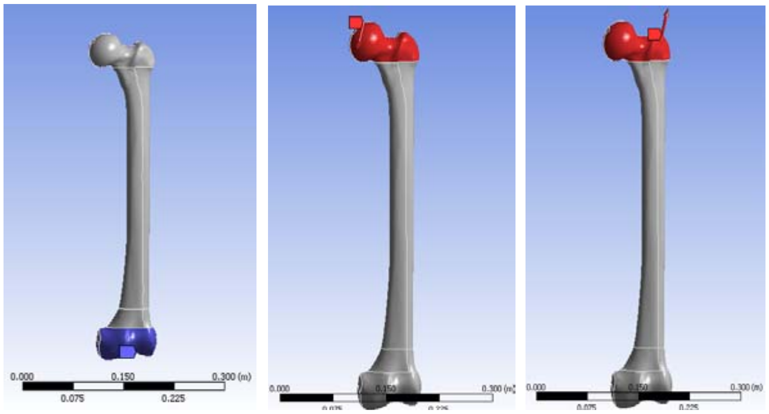
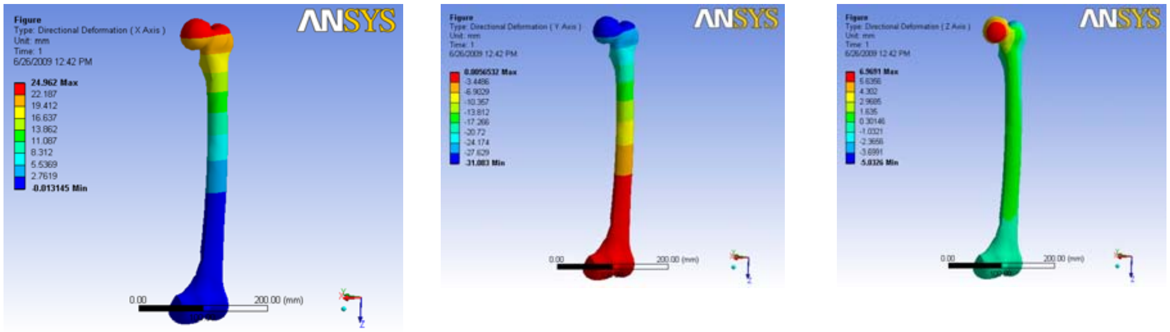


Figure 4. Imposing fixed support and applying forces on the femoral head

Table 2. Force values

Object Name	*Fixed Support*	*F1*	*F 2*
State	Fully Defined		
Scope			
Scoping Method	Geometry Selection		
Geometry	2 Faces	1 Face	
Definition			
Type	Fixed Support	Force	
Suppressed	No		
Define By		Components	
X Component		785. N (ramped)	-305. N (ramped)
Y Component		-589. N (ramped)	0. N (ramped)
Z Component		2158. N (ramped)	-1138. N (ramped)

Based on Finite Element Analysis we determined which areas suffer deformations. This analysis verifies the deformations along x, y and z axes (figure 5). Total deformation obtained is shown in figure 6.



X axis deformation *Y axis deformation* *Z axis deformation*

Figure 5. Directional deformations

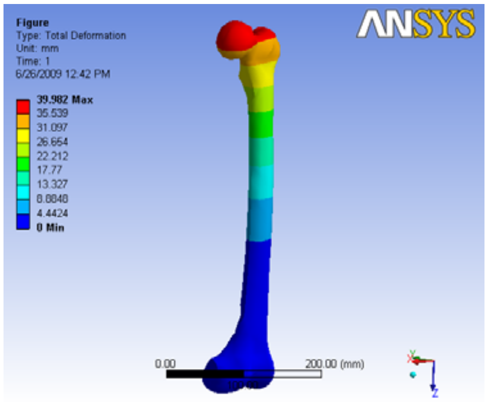


Figure 6. Total deformation of the femur under the action of forces F1 and F2

From ANSYS report, the results of the deformation analysis are presented in table 3, underlying minimum and maximum values for each one.

Table 3. Minimum and maximum deformations

Object Name	*Total Deformation*	*Directional Deformation*	*Directional Deformation 2*	*Directional Deformation 3*
State	Solved			
Scope				
Geometry	All Bodies			
Definition				
Type	Total Deformation	Directional Deformation	Directional Deformation	Directional Deformation
Display Time	End Time			
Orientation		X Axis	Y Axis	Z Axis
Results				
Minimum	0. mm	-1.3145e-002 mm	-31.083 mm	-5.0326 mm
Maximum	39.982 mm	24.962 mm	5.6532e-003 mm	6.9691 mm

Next, the analysis considered the normal stress areas σ (x, y, z) by which Von Misses equivalent stress is determined, expressed both numerical (table 4) and in spatial distribution (figure 7).

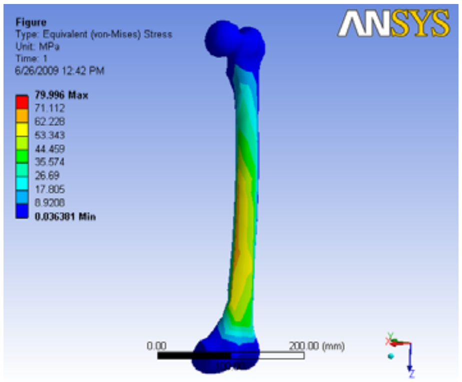
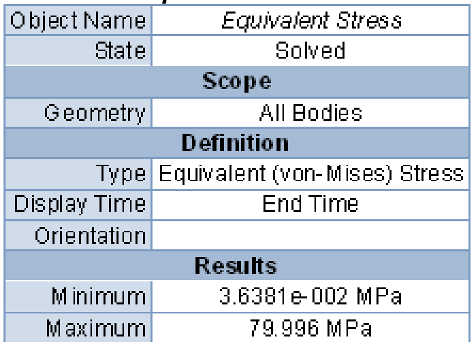


Figure 7. Von Misses equivalent stress diagram

Table 4. Equivalent stress values



Object Name	*Equivalent Stress*
State	Solved
Scope	
Geometry	All Bodies
Definition	
Type	Equivalent (von-Mises) Stress
Display Time	End Time
Orientation	
Results	
Minimum	3.6381e-002 MPa
Maximum	79.996 MPa

3.2. Numerical analysis of the femur-short stem implant assembly

As first step of this analysis, the femur-short stem implant assembly was imported into ANSYS software (figure 8). Contact regions are established between components and proceed to meshing the assembly.

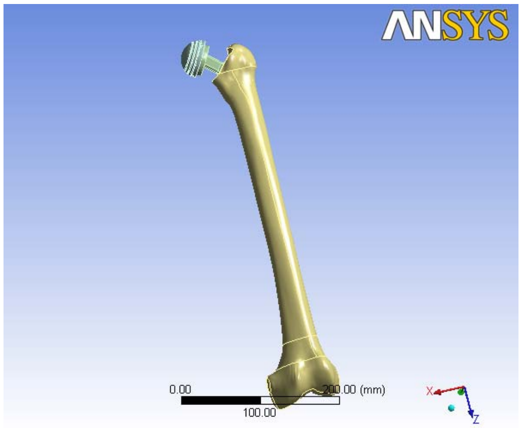
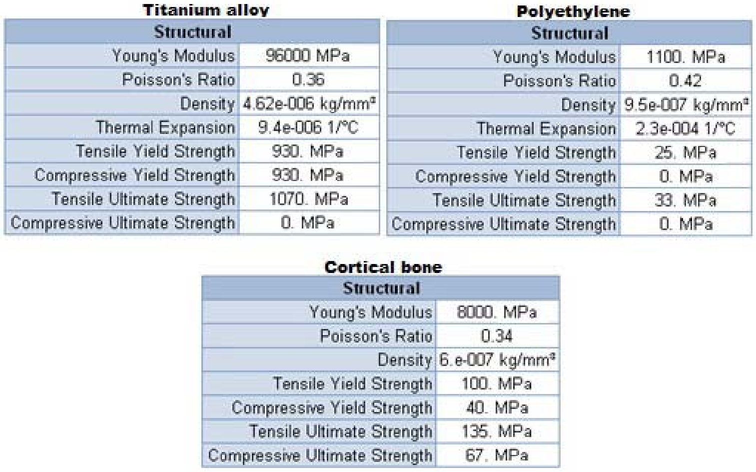


Figure 8. Femur-implant assembly imported in ANSYS

In order to perform the numerical analysis, materials and their properties for each component of the assembly have been chosen (table 5). Titanium alloy and polyethylene were selected for implant materials and the properties of cortical bone were used for the femur.

Table 5. Material properties: titanium alloy/polyethylene/cortical bone



Titanium alloy

Structural	
Young's Modulus	96000 MPa
Poisson's Ratio	0.36
Density	4.62e-006 kg/mm³
Thermal Expansion	9.4e-006 1/°C
Tensile Yield Strength	930. MPa
Compressive Yield Strength	930. MPa
Tensile Ultimate Strength	1070. MPa
Compressive Ultimate Strength	0. MPa

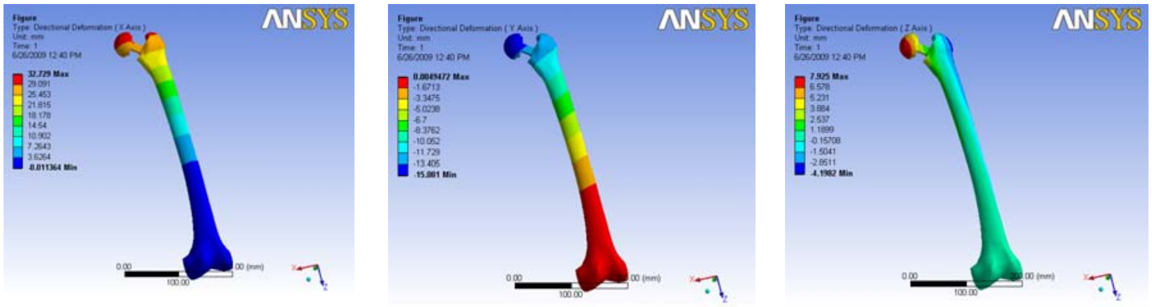
Polyethylene

Structural	
Young's Modulus	1100. MPa
Poisson's Ratio	0.42
Density	9.5e-007 kg/mm³
Thermal Expansion	2.3e-004 1/°C
Tensile Yield Strength	25. MPa
Compressive Yield Strength	0. MPa
Tensile Ultimate Strength	33. MPa
Compressive Ultimate Strength	0. MPa

Cortical bone

Structural	
Young's Modulus	8000. MPa
Poisson's Ratio	0.34
Density	6.e-007 kg/mm³
Tensile Yield Strength	100. MPa
Compressive Yield Strength	40. MPa
Tensile Ultimate Strength	135. MPa
Compressive Ultimate Strength	67. MPa

There are considered the same fixing and loading conditions as in previous case. Thus, the deformations along x, y, z axes (figure 9) and the total deformation (figure 10) of the femur-implant assembly were determined. The results obtained after the analysis are shown in tables 6 and 7.



X axis deformation *Y axis deformation* *Z axis deformation*

Figure 9. Directional deformations X, Y, Z axes

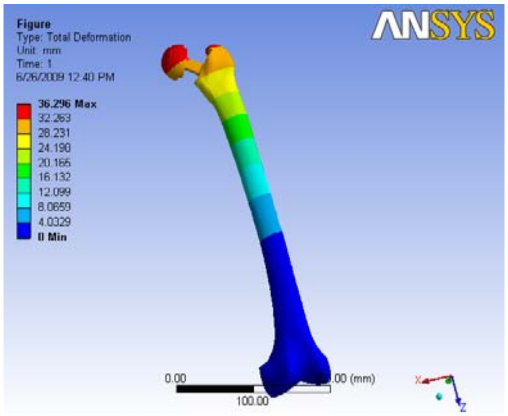
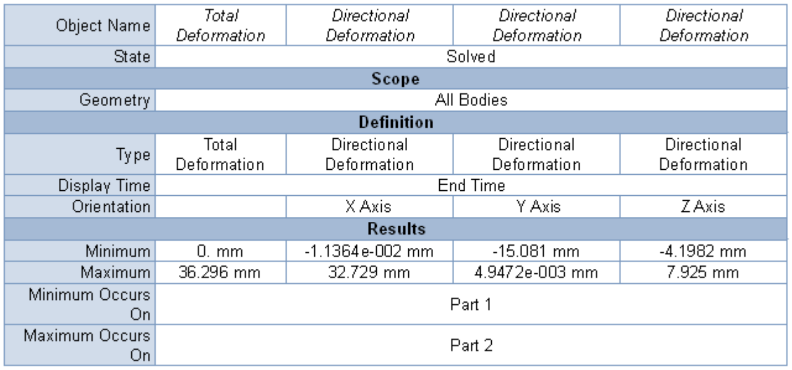


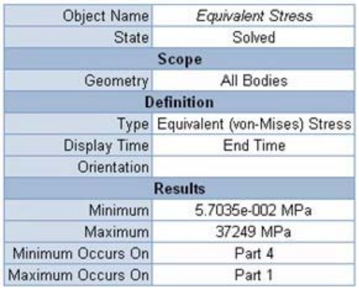
Figure 10. Total deformation of the implanted femur

Table 6. Minimum and maximum deformations



Object Name	*Total Deformation*	*Directional Deformation*	*Directional Deformation*	*Directional Deformation*
State	Solved			
Scope				
Geometry	All Bodies			
Definition				
Type	Total Deformation	Directional Deformation	Directional Deformation	Directional Deformation
Display Time	End Time			
Orientation		X Axis	Y Axis	Z Axis
Results				
Minimum	0. mm	-1.1364e-002 mm	-15.081 mm	-4.1982 mm
Maximum	36.296 mm	32.729 mm	4.9472e-003 mm	7.925 mm
Minimum Occurs On	Part 1			
Maximum Occurs On	Part 2			

Table 7. Results for equivalent stress analysis



Object Name	*Equivalent Stress*
State	Solved
Scope	
Geometry	All Bodies
Definition	
Type	Equivalent (von-Mises) Stress
Display Time	End Time
Orientation	
Results	
Minimum	5.7035e-002 MPa
Maximum	37249 MPa
Minimum Occurs On	Part 4
Maximum Occurs On	Part 1

4. CONCLUSIONS

The performed numerical analysis provides information about biomechanical behavior of a short stem hip implant. Based on this analysis, the distributions of equivalent stress and deformations that occur both in the femur (non-implanted and implanted) and implant were determined.

Materials used for implants must have good mechanical and biocompatibility properties. These properties influence the biomechanical behavior of the implant, both for short and long term.

Further studies are required in order to increase the reliability of the results, considering bone structure and a finer meshing.

5. REFERENCES

[1] Ch. Delaunay: *The Charnley total hip replacement. The Gold Standard of primary hip replacement, 36 years on*, Maîtrise Orthopédique. Le Journal Orthopédique sur le web, http://www.maitrise-orthop.com/corpusmaitri/orthopaedic/mo83_delaunay/delaunay_us.shtml.

[2] D.Y. Jung, S. Tsutsumi, R. Sekel, R. Frost, et al., A New Designed Acetabular Cup-Initial Strength Evaluation Using Finite Element Analysis, Proceeding of the IASTED International Conference BIOMECHNICS, 22-27, 2003.

[3] R. Huiskes, E.Y.S. Chao, A survey of finite element analysis in orthopedic biomechanics: The first decade, *J. Biomechanics*, Vol. 16, No.6, pp. 385-409, 1983

[4] M.S. Kulkarni, S.R. Sathe, K.C. Sharma, K.H. Sancheti, Finite Element Analysis of Mechanical Behavior of SMP Hip Joint Implanted in Femur Bone, *Trends Biomater. Artif. Organs, Vol 23.(1)., pp 10-15 , 2009*

[5] I. Watt, S. Boldrik, E. van Langelaan, R. Smithuis: *Hip - Total Hip Arthroplasty. Normal and abnormal imaging findings*, The Radiology Assistant, the educational web site of the Radiological Society of the Netherlands, Available at: http://www.radiologyassistant.nl/en/. Accessed: 2009-11-15