MSc Biomedical Engineering

University of Patras

Prof.: Associate Pr. K. Moustakas, PhD Candidate D. Stanev

Student: Elissaios Petrai

petrai@ceid.upatras.gr

Modeling and Simulation

FINITE ELEMENT ANALYSIS OF FEMORAL NECK FRACTURES

2018

1. Introduction

Finite element analysis, unlike conventional analysis methods, can be used to analyze structures of complicated geometry and inhomogeneous material properties, making it an ideal tool for analyzing living tissue such as bone, which has complicated geometric shapes and inhomogeneous material properties. In short, FE analysis is a computer based numerical analysis method which can be used to calculate the response of a model to a set of well-defined boundary conditions [1].

Femoral neck fracture is one of the most common injuries in the elderly patients, who are both more likely to have unsteadiness of gait and reduced bone mineral density, e.g. for example people with osteoporosis, predisposing to fracture. Beside its commonness, treatment of these fractures make great challenges to orthopedic surgeons [2]. In the present study, we used finite element models of the proximal femur to assess the mechanical stress and strain of the femoral neck during a gain movement and evaluate the fracture risks factors.

Demographic estimations in Germany, have shown that neck fractures incidence is continuously increasing. Until the year 2020, a rise of 74% in prevalence of proximal femoral fractures is predicted to occur. Especially in the aging patients, femoral neck fractures characterize a significant health problem [2]. Femoral neck fracture is a common fracture disease which accounts for 3.6 percent of the whole-body fracture.

Osteoporosis which includes bone mass loss and the decrease of bone density is an important pathogenic factor of femoral neck fracture. Thus femoral neck fracture is always called "pathological fracture" [3]. Osteoporosis is a type of disease that effect's most of the post menopausal woman but it is also found in men. Osteoporosis occurs in a condition when bone loses minerals such as calcium and phosphate. This results in the lower body mass density [3], [4] .

In this study we create three different finite element model of the femur using FEBio Suite. One model considering a healthy person, one considering a person with lower bone density, and another one considering an person suffering from osteoporosis.

2. MATERIAL AND METHODS

Muscle and hip-joint reaction forces during a gait cycle, were calculated using the musculoskeletal model which was provided. Those values were applied to the finite-element model to obtain estimates of bone formation and risks of fracture. The estimation of join reaction forces was performed using the open-source musculoskeletal modeling environment called OpenSim. Finite element analysis was performed using FEBio Suite which is consist of three different software (PreView, FEBio, PostView).

In Figure 1 method workflow is represented.

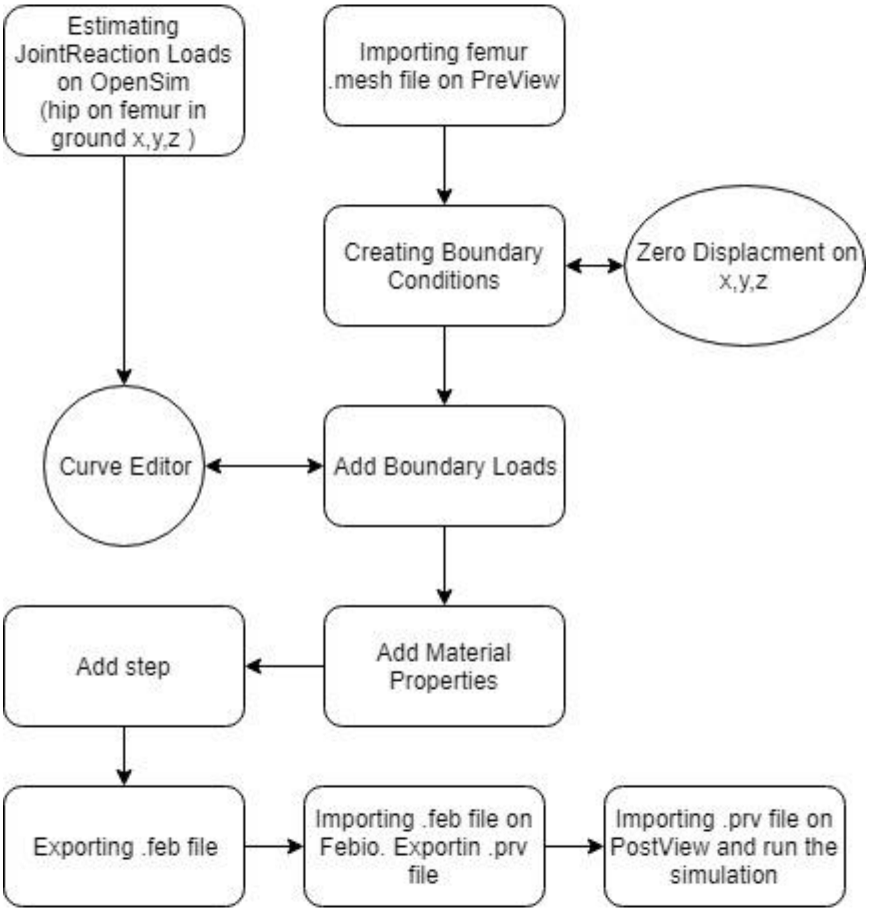

Figure 1 -Methods Workflow

2.1. Femur

Femur is consider to be the longest and largest bone in human anatomy [5] with an average of 48 centimeters (18.9 inch) in length and 2.84 centimeters (1.12 inch) in diameter at the mid shaft for an adult male. Femur it can support up to 30 times the weight of an adult and it has been capable of resisting compression forces of 800-1100kg [1].

It consists of a head and a neck proximally, a diaphysis (or shaft) and two condyles (medial and lateral) distally. The diaphysis of femur is a simplistic, cylindrical structure, while the proximal femur is irregular in shape, consisting of a spherical head, neck and lateral bony protrusions termed the greater and lesser trochanters (Figure 2). The trochanters serve as the site of major muscle attachment. The lateral location of these structures offers a mechanical advantage to assist with abducting the hip [5] .

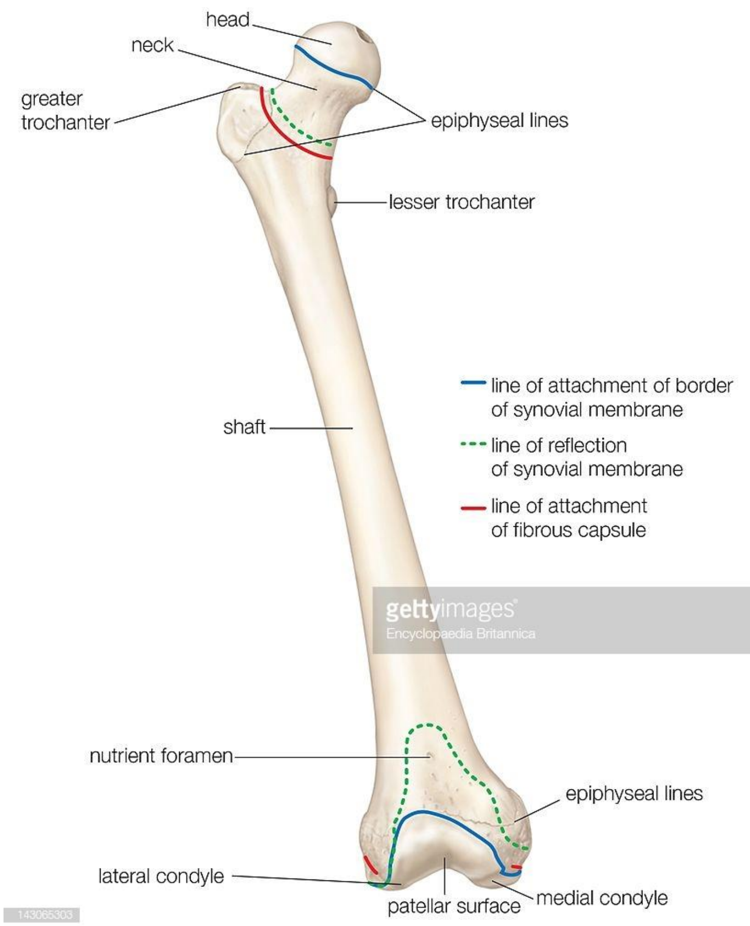

Figure 2 - Femur Bone Anatomy

In the proximal femur are presented both cortical (compact) bone and trabecular (cancellous, spongy) bone. The hard-outer layer of bones is composed of cortical bone tissue. This tissue gives bones their smooth, white, and solid appearance, and accounts for 80% of the total bone mass of an adult skeleton. Its porosity varies from 5% to 10% and its pores consist of space as shown in Figure 3.

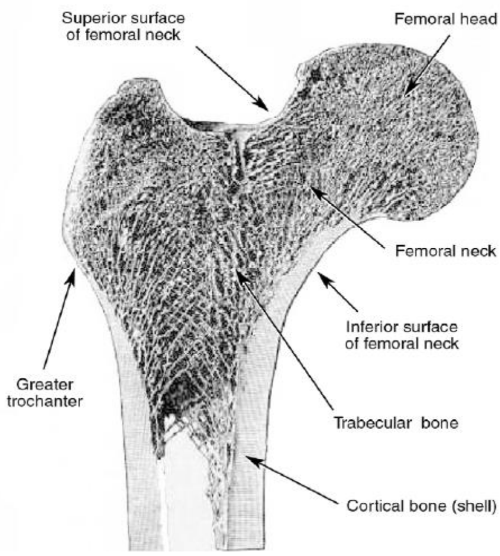

Figure 3 - Anatomy of Proximal Femur

Femoral neck fracture it denotes a fracture adjacent to the femoral head in the neck between the head and the greater trochanter. These fractures have a propensity to damage the blood supply to the femoral head, potentially causing avascular necrosis [5].

Kajzer et al. in their study have indicated that maximum mass of the bone tissue is available at an age of 30 years [6]**.** After that, the phenomenon of bone loss begins to appear. At the age of 45, the loss accelerates significantly, and the rate is about 1~2 percent. When the age is 60, the femoral mass's downward descent appears to be leveling off. Then, the bone loss begin to be followed by a uniform phase [3].

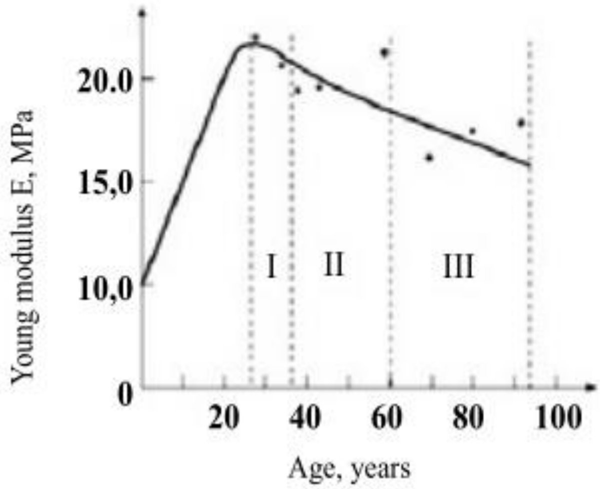

Figure 4 - Dependence between Youngs modulus and human age

2.2. Joint Reaction Analysis on OpenSim

The estimation of join reaction forces, was performed using the open-source musculoskeletal modeling environment called OpenSim.. First, muscle and hip-joint reaction forces were calculated using the musculoskeletal model “gait2396” and the gait dataset which were provided.

The steps in OpenSim:

a. File→ Open Model→ ../path/subject01_simbody_adjusted.osim

b. Tools → Static Optimazation → Load → ../path/setup_static_optimization. Three new files are created in results folder.

c. Tools → Analyze → Load → ../path/setup_static_optimization. In tab “analysis”, we delete “StaticOptimization” and we add “JointReaction”. Then we edit this tab adding step_interval = 10 and giving the full path in the filed “forces_file” where the results we be saved..

After the analysis of joint reaction forces was completed successfully, results for "hip on femur in ground" were plotted as shown in Figure 5. Those results important for boundary conditions.

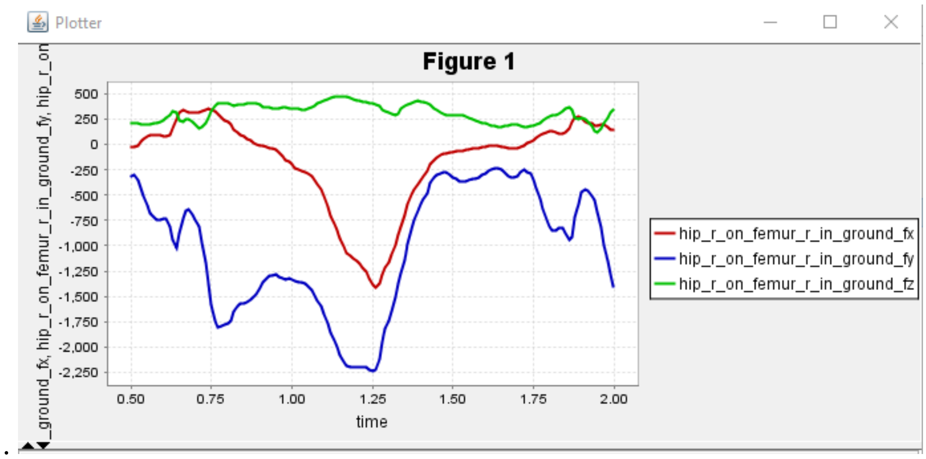

Figure 5 - Joint Reaction Forces "hip on Femur in ground x,y,z"

2.3. Developing Finite Element Models

Finishing the analysis in OpenSim, the next step was to create the finite element models. The .mesh file was imported in PreView as shown in Figure 6.

2.4. Boundary Conditions

Boundary conditions are important aspect of elements analysis, because they affect the obtained results and they include constraints and loads. In the present model muscles and ligaments are not included

In order to complete the finite element model in the above step, boundary conditions are needed to be applied. Therefore, the next step was to add boundary conditions for the distal femur considering it with "fixed displacement in axis x,y,z" Figure 7. Then we applied "boundary load" on femur head Figure 8 .

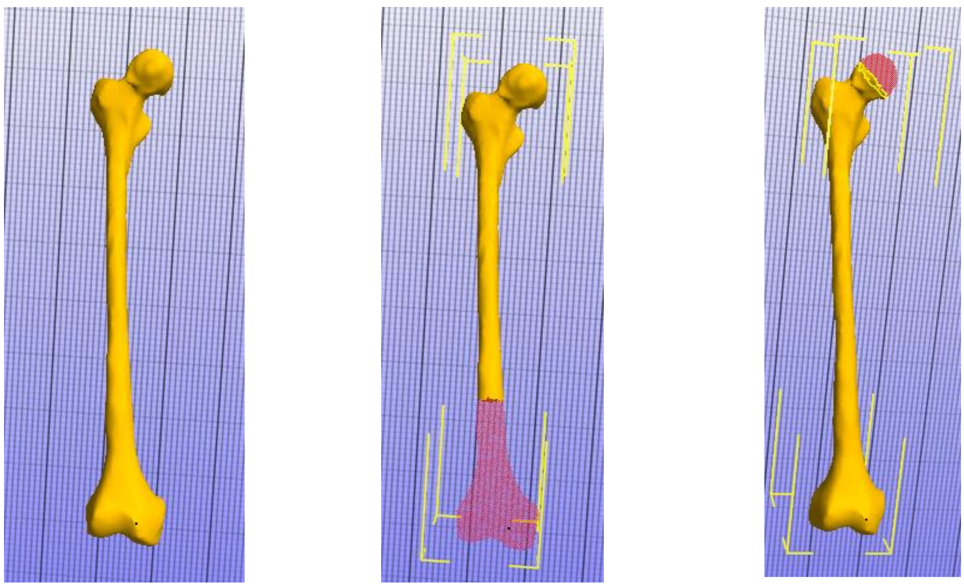

Figure 6 Figure 7 Figure 8

After applied “boundary load” on femur head, we use the Curve Editor of ProView in order to apply the values (Figure 5) that resulted from analysis on OpenSim, as shown in Figures 9-11.

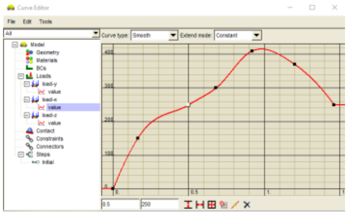
Figure 9

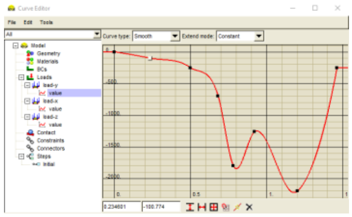
Figure 10

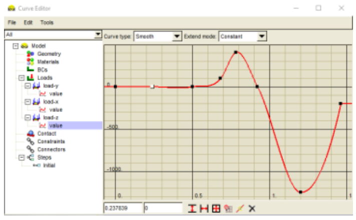
Figure 11

2.5. Material

Most of the research at domestic and abroad considers bone materials as rigid materials, linear-elastic materials and nonlinear-elastic materials, and most studies assume that the whole bone tissue is an isotropic material. Although bone is known to be anisotropic biomaterial, the number of elastic constants is too much, generally it is simplified as orthotropy [3] .

In this study we defined *f*emur as a linear Neo-Hookean material, homogenous and isotropic.

Values of material properties were gathered as a mean value from the literature [4], [7], [3], [8], [9], [10], [11], [12], [13], [14], [15], [16].

Young modulus depend on human age, as shown in Figure 4.

All values selected for our finite element model (Table 1), are within the range of published values:

Table 1 - Material Properties of Cortical Bone used in FE models

	Density (tonne/mm^3)	Young Modulus (MPa)	Poisson’s Ration (MPa)
Normal	1.132e-6	20000	0.3
Normal _lower_density	1e-6	20000	0.3
Osteoporosis	1.132e-6	15000	0.3

Finally with add step, running a dynamic analysis.

Completing those steps, the .feb file was exported from PreView which is an input in FEBio. FEBio is a nonlinear implicit FE solver and the output file .xplt is an input in PostView. Postview is a finite element post-processor that is designed to visualize and analyze results from an FEBio analysis. PostView is the software where we run the simulation.

3. Results and Discussion

3.1. Displacement on YZ axis

In FEBio Suite density does not matter for static analysis. For this reason, we run dynamic analysis.

In another study [3] it has been shown that displacement of femoral head it depends to density conditions. More specifically their results shown that when they decreased density by 10%, then displacement was increased by 11.1 %.

In our study we took results in the same way, smaller the density implies bigger displacement and therefore more possibility for femur neck fracture.

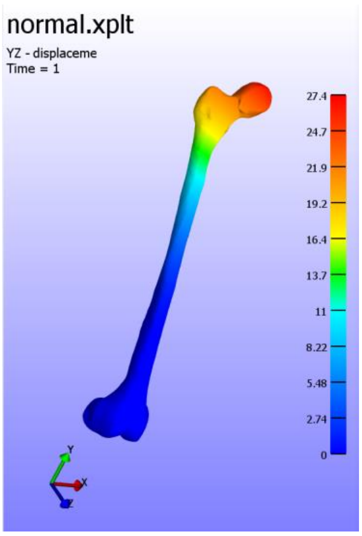

Figure 12

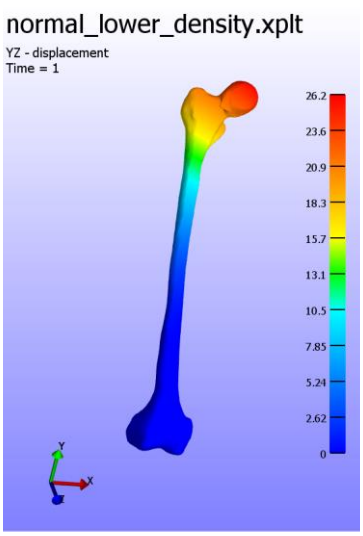

Figure 13

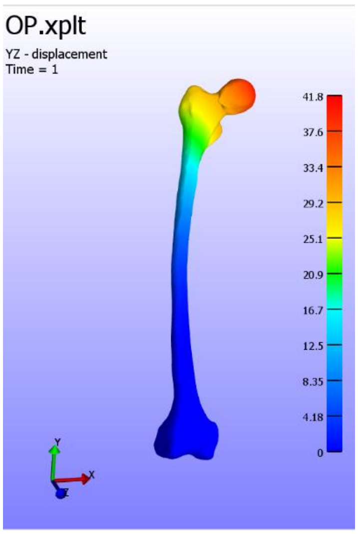

Figure 14

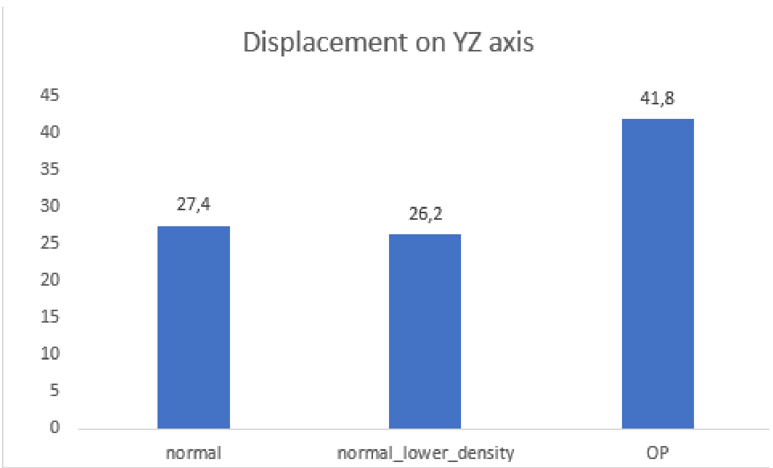

Figure 15

3.2. Maximal Stress Distribution During Walking

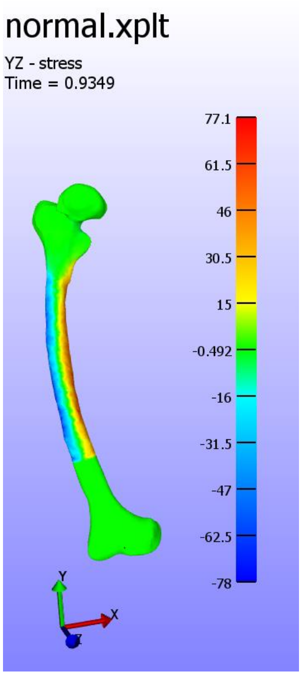

Figure 16

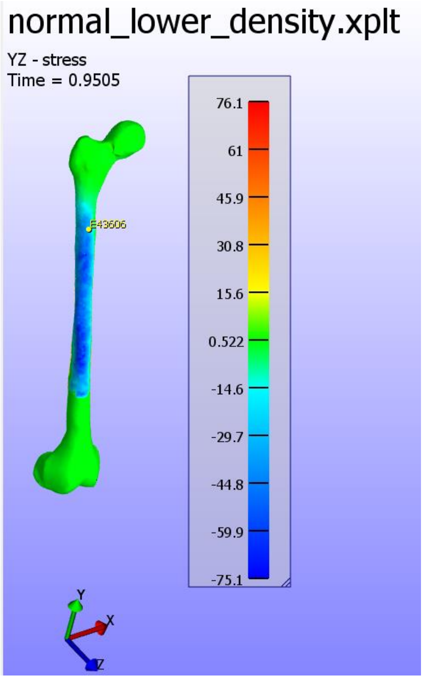

Figure 17

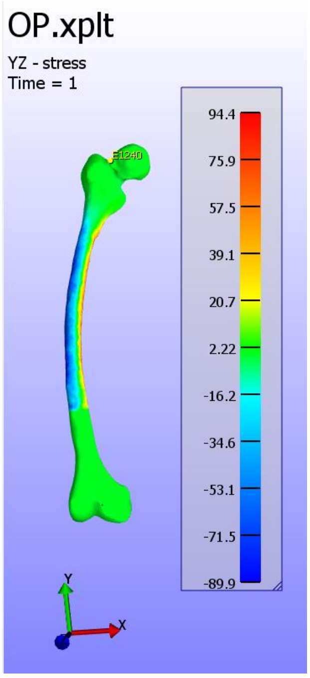

Figure 18

From reaction joint analysis in OpenSim, results that the maximum loading condition on femur bone for our gait model standing on one foot is 2250N (Figure 5). For this maximal loading condition, the maximum pressure appears on model with osteoporosis and is 94.4MPa. For normal model and for model with lower density is 77.1MPa and 76.1MPa respectively.

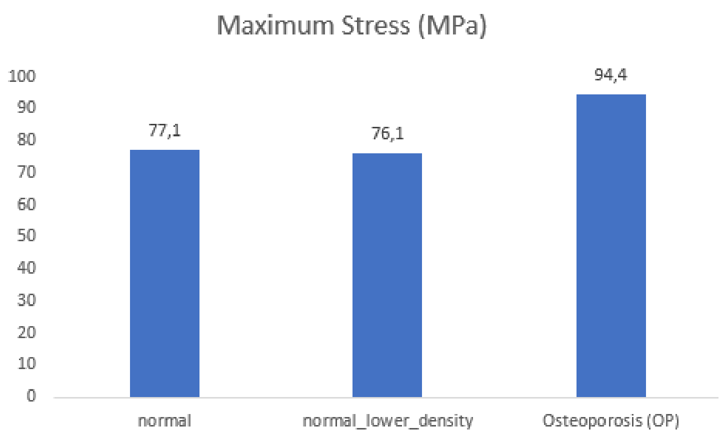

Figure 19

Those results seems to be very close to the results in other study presented [6], where for an maximum applied load of 2317N on femur for a person standing on one foot, the maximum stress acting on the bone was depending between 29.7MPa and 65.6MPa, as shown in the following Table 2. The young modulus of this study, that affects the maximum stress for each person 1 to 10, it depends on Figure 4. Again, very close material values with our study.

Table 2 – Results presented in other study

Name of the patient	Age (years)	Stress (Mpa)
Person 1	27	24.1
Person 2	34	29.7
Person 3	40	37.9
Person 4	43	33.2
Person 5	52	44.5
Person 6	61	51.2
Person 7	67	59.9
Person 8	68	53.1
Person 9	76	60.4
Person 10	84	65.6

The maximal stress is proportional to the maximal strength applied on the femur neck. At this point, for the model with osteoporosis, we choose a random point around the femur neck. As shown in Figure 20 a stress of 25MPa is applied at femur neck at step 8, which it shown the possibility for a femur neck fracture.

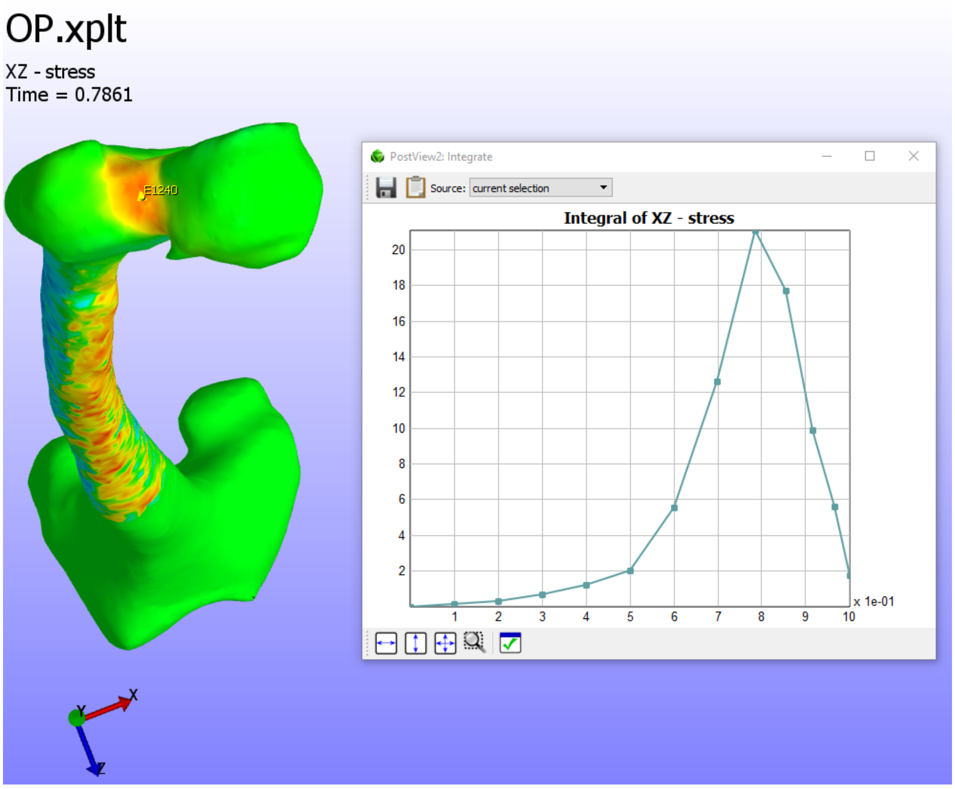

Figure 20 – Stress around specific point of femur neck

3.3. Pressure

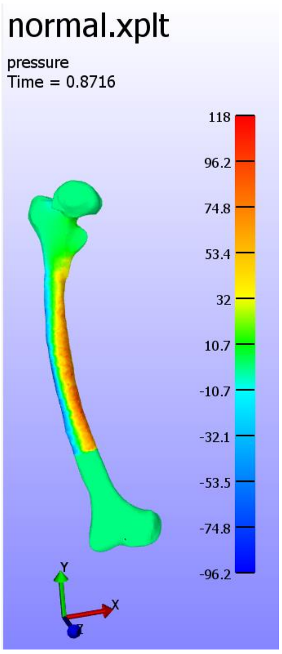

Figure 21

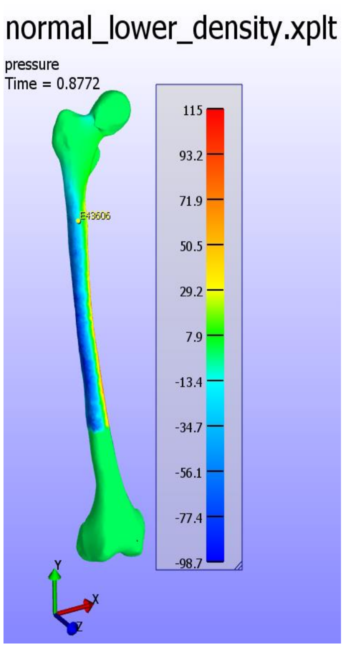

Figure 22

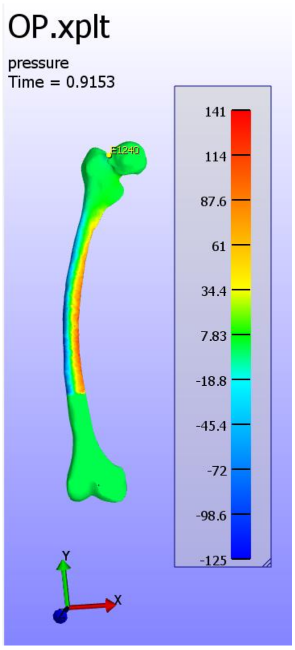

Figure 23

As shown in the above figures, again the model with osteoporosis seems to be affected more than the other two models, with a maximum pressure of 141MPa. Those results are considered to be within the range reasonable values in bibliography.

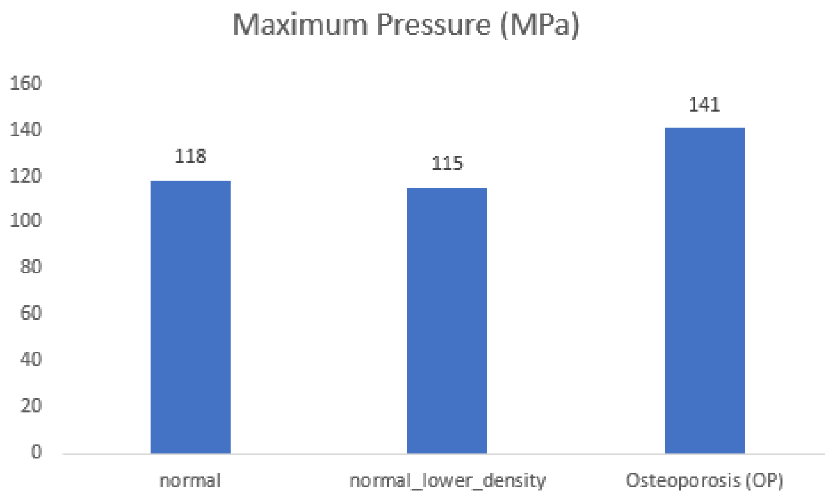

Figure 24

References

[1] R. Nareliya and V. Kumar, "FINITE ELEMENT APPLICATION TO FEMUR BONE: A REVIEW," vol. 3, no. 1, p. 6, 2012.
[2] H. R. S. Hosseinzadeh, M. Qoreishy, and A. S. Shahi, "Finite Element Analysis of Fixation Device for Femoral Neck Fracture: Dynamic Hip Screw," vol. 1, no. 2, p. 8, 2014.
[3] Wang Monan, Zhang Meng, and Xu Wenwen, "Finite element analysis for the influence of bone density on the mechanical properties of femoral neck," in *2010 International Conference on Computer Application and System Modeling (ICCASM 2010)*, Taiyuan, China, 2010, pp. V8-365-V8-368.
[4] M. S. Lahari, Vijay, and M. Anburajan, "Finite element analysis of femur in the evaluation of Osteoporosis," in *2011 3rd International Conference on Electronics Computer Technology*, Kanyakumari, India, 2011, pp. 415–419.
[5] M. B. N. Fanisam, P. Mitesh, and P. Nitin, "Finite Element Analysis of Femur Bone," p. 17, 2017.
[6] M. V. Kumar Reddy, B. Ganesh, and K. Bharathi, "Use of Finite Element Analysis to Predict Type of Bone Fractures and Fracture Risks in Femur due to Osteoporosis," *J. Osteoporos. Phys. Act.*, vol. 4, no. 3, 2016.
[7] R. Schwarzkopf, "Finite Element Analysis of Femoral Neck Stress in Relation to Pelvic Width," p. 6.
[8] M. Noda, Y. Nakamura, K. Adachi, Y. Saegusa, and M. Takahashi, "Dynamic finite element analysis of implants for femoral neck fractures simulating walking," *J. Orthop. Surg.*, vol. 26, no. 2, p. 230949901877789, May 2018.
[9] D. S. Angadi, "Biomechanical analysis of femoral fracture fixation using the expert adolescent lateral femoral nail system," p. 398.
[10] A. Bettamer, "prediction of proximal femur fracture: finite element modeling based on mechanical damage and experimental validation," p. 184.
[11] A. B. Costa, "Multiscale analysis of the hip joint: from body motion to cartilage stresses," p. 42.
[12] A. T. M. Phillips, C. C. Villette, and L. Modenese, "Femoral bone mesoscale structural architecture prediction using musculoskeletal and finite element modelling," *Int. Biomech.*, vol. 2, no. 1, pp. 43–61, Jan. 2015.
[13] C. Deng, "Femur/femoral neck load estimation during stair navigation," p. 130.
[14] A. R. MacLeod, H. Rose, and H. S. Gill, "A Validated Open-Source Multisolver Fourth-Generation Composite Femur Model," *J. Biomech. Eng.*, vol. 138, no. 12, p. 124501, Nov. 2016.
[15] T. A. Brown, L. Kohan, and B. Ben-Nissan, "Assessment by finite element analysis of the impact of osteoporosis and osteoarthritis on hip resurfacing," p. 6.
[16] R. Heydon, "Finite Element Analysis of Knee Articular Cartilage," p. 87.