# Determination of Mechanic Resistance of Osseous Element Through Finite Element Modeling

E. Isaza[1], E. Salazar[1], L. Florez[1]

[1]Universidad Tecnológica de Pereira, Pereira, Risaralda, Colombia

## Abstract

Clinicians, patients and engineers would benefit from accurate methods of predicting the mechanical properties of bone in order to develop better prosthesis, orthesis or other class of devices oriented to help or increase the human well-being.

In this paper, the geometry and dimensions of a human femur was generated through 3D scanning process, trying to obtain the highest quality in the femur external topology. The complex structure of the femur was represented through three dimensional anisotropic finite element model, built with COMSOL Multiphysics®. This model was subjected to various mechanical loads according with experimental reference studies. The reference studies, was developed using dual energy X ray absorptometry (DXA), quantitative computed tomography (QCT) and real human bones, which was subjected to specific loads in order to establish an objective basis for comparison.

The main goal of this study is to model and test a human femur through finite element technique, validating the results by comparison with experimental studies.

This paper presents very approximate results and shows that the finite element model is a promising method for predicting fracture loads and can be used for further study of a mechanical behavior of bones.

COMSOL was used to build a complex model comprising the cortical bone and trabecular bone. The trabecular bone is the material making up the proximal femur, which forms the hip joint. Cortical bone is the material of the neck of the femur, the thinner part of the femur but also the most compact and resistant. The principal challenge in the model building, was the construction of a valid geometry for meshing could adequately represent the combined behavior of both materials.

Among the findings of the study emphasizes the need to represent the human femur as a composite of at least two anisotropic materials. Ignoring this leads to models that are not consistent with reality. COMSOL allowed for a complex model with relative ease, thanks to its meshing options and compatibility with CAD software used in this case, SolidWorks®. Additionally, it should be emphasized that the main vulnerability found in the human femur through the model developed by COMSOL was the neck of the head, which is consistent with empirical studies with real bones. In the final analysis, this study shows that there is great research

perspective using the finite element method combined with the nonlinear mechanical behavior study of bones and many more parts of the body.

## Reference


1. J.M Rodríguez. , et al. “Estudio de Transmisión de Carga en Fémur Humano”. Ingeniería Mecánica Tecnología y Desarrollo. Vol. 2 No. 5 pp. 164-168. (2007)

2. Huskies, Rik. Pascal, H. G, Strens E. “Interface Stress in the Resurfaced Hip. Acta Orthop Scand. Vol 56. pp. 474-478. (1985)

3. Testi et al. “Risk of Fracture en Elderly Patients: A New Predictive Index Based on Bone Mineral Density and Finite Element Analysis.” Computer Methods and Programs in Biomedicine 60. pp. 23-33. 1999.

4. Cernescu, A. “Reverse Engineering and FEM Analysis for Mechanical Strength Evaluation of Complete Dentures: A Case Study.” Reverse Engineering-Recent Advances and Applications. Edited by: Alexandru C. Telea. Published by: Intech. pp. 263-276. (2012).

5. Chang, Kuang-Hua. “A Review on Shape Engineering and Design Parametrization in Reverse Engineering.” Tomado de: “Reverse Engineering-Recent Advances and Applications.” Edited by: Alexandru C. Telea. Published by: Intech. pp. 161-186. (2012).

6. Cody Dianna D et al.”Femoral Strength is Better Predicted by Finite Element Models Than QCT and DXA.” Journal of Biomechanics 32. pp 1013-1020. 1999.

7. In Gwun Jang, Il Yong Kim, "Application of design space optimization to bone remodeling simulation of trabecular architecture in human proximal femur for higher computational efficiency. Finite Elements in Analysis and Design. Volume 46. pp. 311–319. (2010)

8. Ridha Hambli. "Numerical procedure for multiscale bone adaptation prediction based on neural networks and finite element simulation". Finite Elements in Analysis and Design. Volume 47 pp. 835–842 (2011).

9. Sharon M. Swartz, Antigone Parker, Christine Huo. Theoretical and Empirical Scaling Patterns and Topological Homology in Bone Trabeculae. The Journal of Experimental Biology Volume 74B, 573–590 (1998).

10. P.J. Rubin et al. The Mophology of the Proximal Femur. The Journal of Bone and Joint Surgery. pp. 20-32 (1992).

11. C.M. Muller-Karger et al. p-version of the finite-element method for highly heterogeneous simulation of human bone. Finite Elements in Analysis and Design. Volume 40. pp. 757-770 (2004)

## Figures used in the abstract

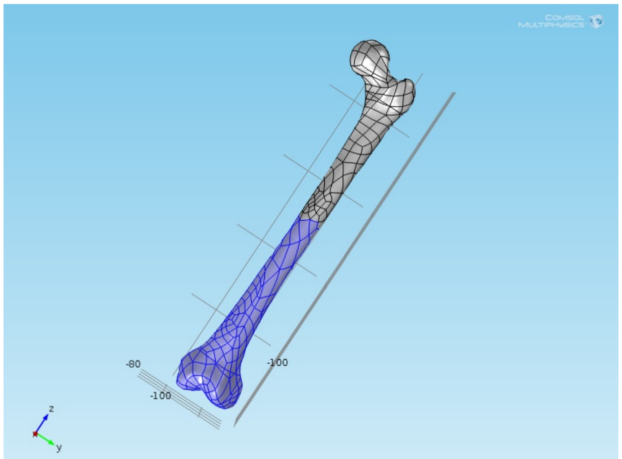


**Figure 1**: Bone Fixing for Simulation.

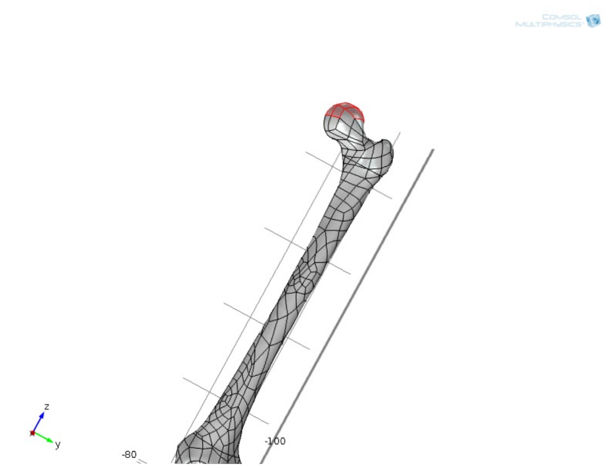


**Figure 2**: Loads on Bone.

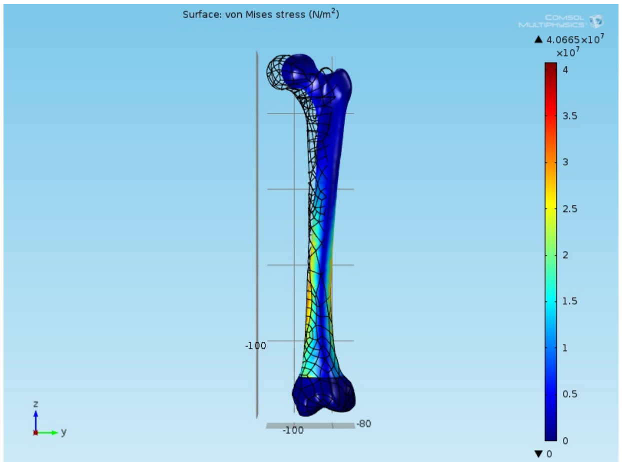


**Figure 3**: Results: Bone Stress Distribution.

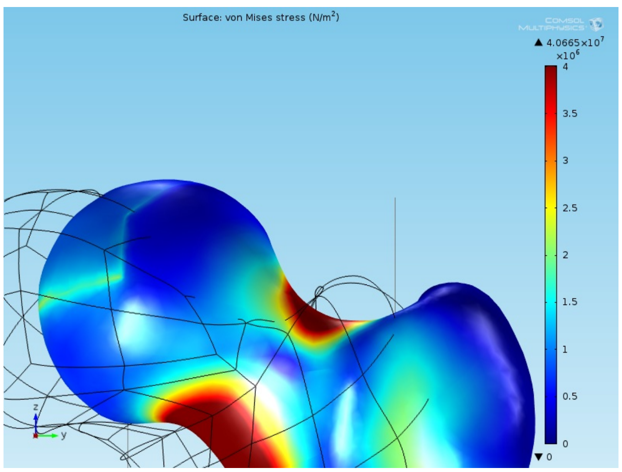


**Figure 4**: Head Stress Distribution.