15th International Symposium on Computer Methods in Biomechanics and Biomedical Engineering
and
3rd Conference on Imaging and Visualization
CMBBE 2018
P. R. Fernandes and J. M. Tavares (Editors)

Abstract ID-202

HEXAHEDRAL FINITE ELEMENT MESH GENERATION FOR TOTAL HIP ARTHROPLASTY ANALYSIS

Diogo F. Almeida[(1)], Rui B. Ruben[(2)], João Folgado[(3)], Paulo R. Fernandes[(3)], Benedict Verhegghe[(4)], Matthieu De Beule[(4)]

[(1)]**IDMEC, Instituto Superior Técnico, Universidade de Lisboa; iBiTech-bioMMeda, Ghent University, Portugal**
diogo@dem.ist.utl.pt

[(2)]**ESTG, CDRSP, Polytechnic Institute of Leiria, Portugal**
rui.ruben@ipleiria.pt

[(3)]**Universidade de Lisboa, Portugal**
jfolgado@tecnico.ulisboa.pt, paulo.rui.fernandes@tecnico.ulisboa.pt

[(4)]**iBiTech-bioMMeda, Ghent University, Belgium**
benedict.verhegghe@ugent.be, matthieu.debeule@ugent.be

Keywords: Mesh Generation, Image Based Simulation, Hip implants

Summary: In the last decade, our group has been working in the development of a new software for total hip arthroplasty (THA) preparation. From a CT-scan, the developed computational toolbox is able to create a 3D femur model, extract important landmarks, propose an optimal prosthesis placement and compute important performance criteria. This toolbox is fully automatic, i.e., no human intervention is necessary. However, a user interface is provided for the doctor to test or even to change the software's suggestion. Automatic mesh generation for THA finite element (FE) analysis is one of the last steps of the toolbox. With a finite element analysis, the toolbox can compare some performance criteria between several prostheses. The developed toolbox is able to work with low-resolution CT-scans and also with low-dose CT-scan protocols.

Since, hexahedral meshes present several numerical advantages over tetrahedral meshes, a method for automatically generate hexahedral FE meshes of the femur-implant coupling was implemented. Contrarily to prostheses with rectangular cross-sections, where the meshing is straightforward, the meshing of the prostheses with round cross-sections is done combining two techniques: grid-based and receding front. Implants have a smooth geometry. However, femur has a non-smooth exterior surface. To reduce elements distortion an optimization procedure was also implemented to rearrange femur tissue nodes.

In the interface, nodes of the implant are made coincident with the femur tissue nodes, in order to increase contact analysis quality. To each element of the mesh of the femur there is a type of material associated, medullary cavity, cortical or trabecular bone, depending on location. Every section has the same number of quadrilateral elements and therefore the hexahedral elements are formed by the consecutive quadrilateral elements. There is the possibility of interactively choosing the number of elements per section and the number of sections along the implant, increasing or decreasing the mesh refinement. The condition number of the Jacobian matrix was used to compute the FE mesh quality. To illustrate the automatic mesh generation procedure, a comparative FE analysis was successfully performed in order to compute important THA performance criteria.