PENNSTATE
6th OpenFOAM® Workshop
PennState University, USA
13-16 June 2011

CONTACT STRESS ANALYSIS IN OPENFOAM: APPLICATION TO HIP JOINT BONES

Philip Cardiff*†[1], Alojz Ivanković[1], David FitzPatrick[1], Rob Flavin[1], and Aleksandar Karač[2]

[1]School of Electronic, Electrical and Mechanical Engineering, University College Dublin, Belfield, Dublin, D4, Ireland
[2]University of Zenica, Zenica, Bosnia and Herzegovina

March 11, 2011

contact stress analysis, hip joint, finite volume, finite element.

1 INTRODUCTION

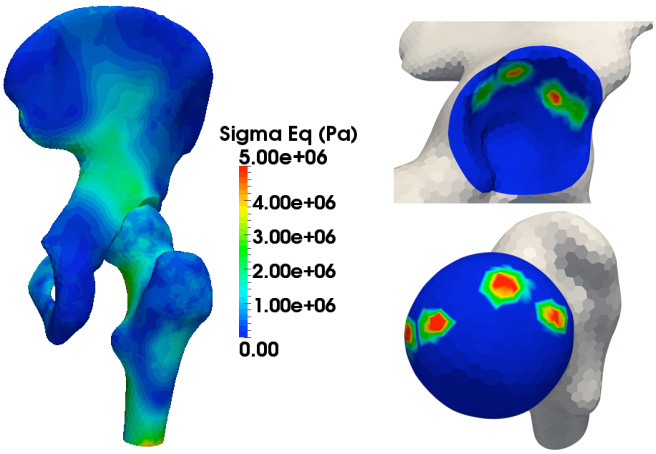
Total Hip Arthroplasty is a surgical procedure that reforms the hip joint, replacing the pathological joint with an artificial prosthesis. Due to post-operative joint instability, complications such as dislocation are still a significant problem [1]. This research aims to develop a realistic numerical model of a healthy hip joint and examine its stability. An initial model of the hip joint simulating stance has been developed using a *homogenous* material model.

2 METHODS & RESULTS

A 23-year-old male subject was chosen with no congenital or acquired pathology of the hip joint. Computed tomography (CT) and magnetic resonance imaging (MRI) scans of the subject's hip joint were acquired. The 3D bone surface geometry was extracted from the scans using open-source software 3D Slicer. Volume meshing was performed using Ansys ICEM CFD and the OpenFOAM utility *polyDualMesh*. A finite volume contact stress solver implemented in OpenFOAM was employed. Initially a *homogenous* model was considered; the bone was assumed to be a homogenous isotropic linear-elastic material with a Young's modulus of 500 MPa and a Poisson's coefficient of 0.2 [3].

The stance phase of gait has been simulated; a force corresponding the ground reaction force experienced by the hip during the stance phase of the gait analysis was applied to the distal end of the femur. The pelvis was fixed at the iliopubic joint and the iliosacral joint. Maximum

†Corresponding Author: Philip Cardiff (philip.cardiff@ucd.ie)



(a) Sigma Equivalent (Pa)

(b) Contact pressure (areas of high contact pressure are in red)

Figure 1: OpenFOAM *homogenous* hip joint model stress results

contact pressures of 10 MPa were found on the articular surfaces. A maximum sigma equivalent stress of 8 MPa was observed at the iliosacral joint fixture. The OpenFOAM *homogenous* model results were verified against finite element software Abaqus and good agreement was obtained. A *sandwich* model is currently being developed which assumes bone to be composed of a flexible cancellous core sandwiched inside a stiffer cortical shell.

3 CONCLUSIONS

A procedure has been developed to create a patient-specific 3D hip joint model from CT and MRI scans. Numerical analysis was performed using finite volume based OpenFOAM and compared with finite element based Abaqus. Initial results agree well with literature [3, 4]. The authors wish to acknowledge financial support from Tekno-Surgical, and the SFI/HEA Irish Centre for High-End Computing (ICHEC) for the provision of computational facilities.

References

[1] R. D. Beckenbaugh and D. M. Ilstrup. Total hip arthroplasty. *The Journal of Bone and Joint Surgery*, **60**, 306-313, 1978.

[2] F. Bachtar, X. Chen and T. Hisada. Finite element contact analysis of the hip joint. *Medical & biological engineering & computing*, **44**(8), 643-51, 2006.

[3] M. Dalstra, R. Huiskes, A. Odgaard and L. Van Erning. Mechanical and textural properties of pelvic trabecular bone. *Journal of biomechanics*, **26**(4-5), 523-535, 1993.

[4] M. Daniel. Contact stresses in the human hip joint - review. *Journal of Biomechanics*, **2**(8), 197-203, 2006.

[5] A. E. Anderson. Computational modeling of hip joint mechanics. *PhD Thesis*, University of Utah, 2007.