STRUCTURAL ANALYSIS OF METALLIC ORTHOPAEDIC IMPLANTS BASED ON ASTM STANDARDS: A SOFTWARE COMPARISON STUDY

Federico Guidolin and Luca Modenese

Graduate School of Biomedical Engineering, University of New South Wales, Australia

Introduction

The finite element (FE) method is a well-established method for performing *in silico* structural analysis of orthopaedic implants. In practice, this approach involves the generation of a volumetric mesh from a CAD object representing the implant, used to define the FE model (choosing element type and order, plus defining of material properties, boundary conditions and loads). The FE model is then solved by specialised software packages, producing estimations of material stresses and strains that can be further analysed. The FE analysis of orthopaedic implants is regulated by the American Society for Testing and Materials (ASTM) through multiple standards, e.g. [1-2].

In terms of tools available for performing these analyses, in recent years there has been a general trend towards the development of high-quality open-source computational packages aimed to the biomechanical community, including FEBio [3], a finite element solver integrated with pre- and post-processor utilities. Despite FEBio was verified in multiple test cases against Abaqus and NIKE3D [3], we are not aware of any comparison against commercial software packages for a realistic case like the analysis of an orthopaedic implant. Therefore, the aim of this work is to quantify the agreement between structural analyses performed with FEBio and ANSYS (https://www.ansys.com) following the ASTM standards.

Methods

We applied two workflows for the preparation of the FE model geometry from the CAD model of a femoral component of a knee implant provided with the [1] (retrieved from https://docs.conself.com/validation in STEP format). Our purpose was to compare the software packages considering the various modelling steps. We generated a first tetrahedral mesh (mean element size: 1 mm) directly from the CAD model using Ansys (MESH1). We created a second volumetric mesh (mean element size: 1 mm) transforming the STEP file in STL format with Gmsh [4], improving it using MeshLab [5] and finally using TetGen [6] in FEBio (MESH2). We then imported both meshes in both software packages, defined quadratic elements and applied to the model boundary conditions and loads as defined by ASTM F3161-16. The assigned material was CoCrMo (Young's modulus: 230 GPa, Poisson's coefficient: 0.29), modelled as linear elastic and isotropic. We performed convergence studies to ensure that our estimations of Von Mises stress at the ASTM points of interest (condyle region and notch regions, Figure 1) were reliable in all models and quantified the percent differences between the two software packages and between each software package and the results reported in the ASTM standard.

Results

Table 1 summarises the results from the FE simulations, while Figure 1(B) shows the results of a converged simulation performed in FEBio using MESH2

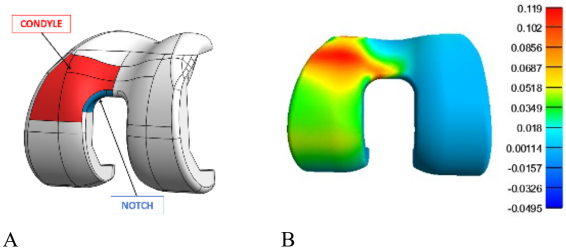

Figure 1: The implant geometry with highlighted points of interest (A) and Von Mises stresses computed by FEBio according to [1] using MESH2.

	MESH1		MESH2	
	condyle	notch	condyle	notch
FEBio vs ASTM	1.8%	9.78%	1.6%	8.2%
ANSY vs ASTM	0%	6.16%	0.2%	8.1%
FEBio vs ANSYS	1.8%	3.8%	1.5%	0.1%

Table 1: Percent differences of Von Mises stresses estimated at the points of interest by FEBio, ANSYS and reported in the ASTM standards.

Discussion

We found that FEBio and Ansys compute similar Von Mises stresses at the points of interest (≈4% maximum difference). Ansys was computationally more efficient and could generate better geometrical models. In fact, we could not generate a usable volumetric mesh directly from the CAD file in FEBio. Pre-processing, however, is not a key FEBio development goal. We are currently performing similar analyses on a hip prosthesis [2] and tibial component to extend and confirm our findings.

References

1. ASTM F3161 – 16, https://www.astm.org/f3161-16.html
2. ASTM F2996 – 20, https://www.astm.org/f2996-20.html
3. Maas et al, J Biomech Eng, 134(1), 2012.
4. Geuzaine et al., Int J Num Meth in Eng, 79:1309-31, 2009.
5. Cignoni et al., Eurograph Ita Chap Conf, 129-136, 2008.
6. Hang, ACM Trans. on Math Soft. 41(11), 2015

Acknowledgements

LM acknowledges UNSW Scientia Fellowship scheme.