NUMERICAL ANALYSIS OF THE TOTAL KNEE JOINT REPLACEMENT - MODELLING IN COMSOL

J. Daněk

Department of Mathematics, University of West Bohemia

Abstract

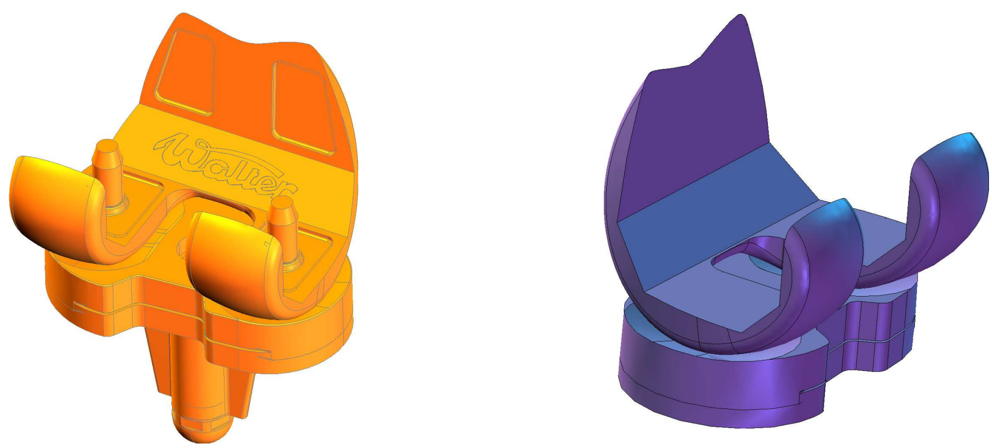
We consider the total knee replacement by MEDIN Orthopaedics company in the paper. The geometry is given by the manufacture's CAD data. The problem is modelled as a contact problem in linear elasticity. The numerical simulation is performed: the replacement is subjected to the load simulating the conditions in the human body; the resulting stress distribution is computed by the COMSOL Multiphysics.

1 Introduction

Aim of the contribution is to provide the distribution of stress and the displacements in all component parts of artificial total knee replacement and to determine the mechanical conditions on the contact interface separating its femoral and tibial parts, such as the load transfer provided by the contact. The results would help in predicting the wear of critical parts of the implant. The results could also be important in analysing the proper positioning of the prosthesis being implanted.

2 The Model

The original CAD description of the prosthesis is uselessly complicated (see the figure 1a). In order that number of finite elements of generated mesh will be suitable for computing we simplify the geometry at the beginning (see the figure 1b).



(a) original CAD geometry (b) simplified CAD geometry

Figure 1: The knee joint replacement by MEDIN Orthopaedics company

The problem is modelled in Structural Mechanics Module in 3D. We suppose that all parts are linear elastic isotropic. The material properties (defined by the Young's modulus E

and Poisson constant ν) are following: for the femur part (alloy $CoCrMo$) $E = 1.15 \times 10^{11}$ [Pa], $\nu = 0.3$, for the plastic spacer (polyethylen UHMWPE) $E = 3.4 \times 10^{8}$ [Pa], $\nu = 0.4$ and for the tibial part (alloy $Ti6Al4V$) $E = 1.15 \times 10^{11}$ [Pa], $\nu = 0.3$. Between the femoral and the tibial part of the prosthesis a contact interface is considered. The tibial component of the replacement is held fixed by prescribing a zero displacements on its bottom part; on the upper parts of the femoral component the load acting on the prosthesis is given such that the biomechanical conditions in the human body are simulated.

The CAD Import Module allows the transition from geometric designs created with specialized CAD tools to the simulation environment of COMSOL Multiphysics and its modules. Aspiration to generate finite element mesh for the total knee replacement was not successful for our manufacture's CAD data. So that we used the TGrid preprocessor to create the finite element mesh (see the figure 2). Generated mesh includes 6596 nodes and 24575 tetrahedrons. Consequently we imported obtained mesh into COMSOL Multiphysics.

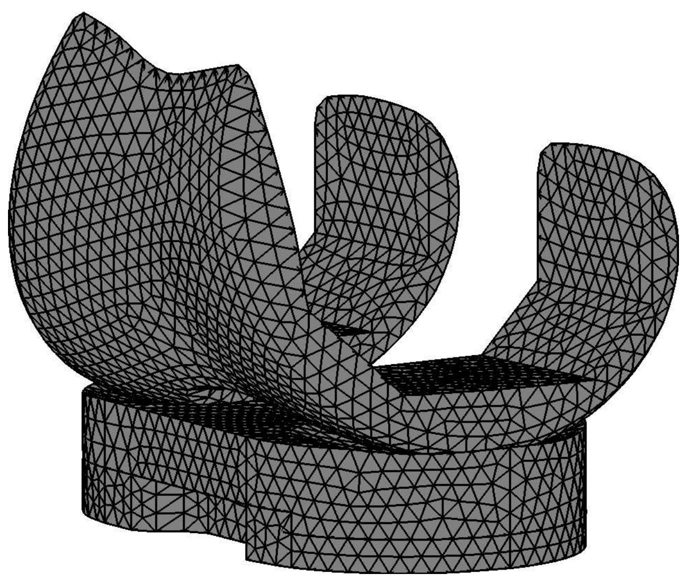


Figure 2: The finite element mesh

3 Numerical results

Number of degrees of freedom is 127173 if we choose default type of element i.e. Lagrange-Quadratic elements. The figures 3 and 4 show total displacement of the total knee replacement. It is clearly to see contact areas between the femoral and the tibial part of the prosthesis on figure 4.

Division of loads acting on the tibial component is of primary importance in orthopedic surgery. Nonobservance of the balance of both compartments (medial and lateral) or possible overloading of the posterior part of the tibia plate leads to wear out of the polyethylene insert. Asymmetrical overloading leads to the premature abrasion of a plastic polyethylen insert with production of a great amount of polyethylene elements, which instigate a complicated inflammatory reaction leading to loosening metallic components of the total replacement from the bone. Von Mises stress is displayed on figure 5.

COMSOL Multiphysics is an useful tool for modelling and simulation of various physical processes and it can help also with the analysis of stress distribution in the joint replacements.

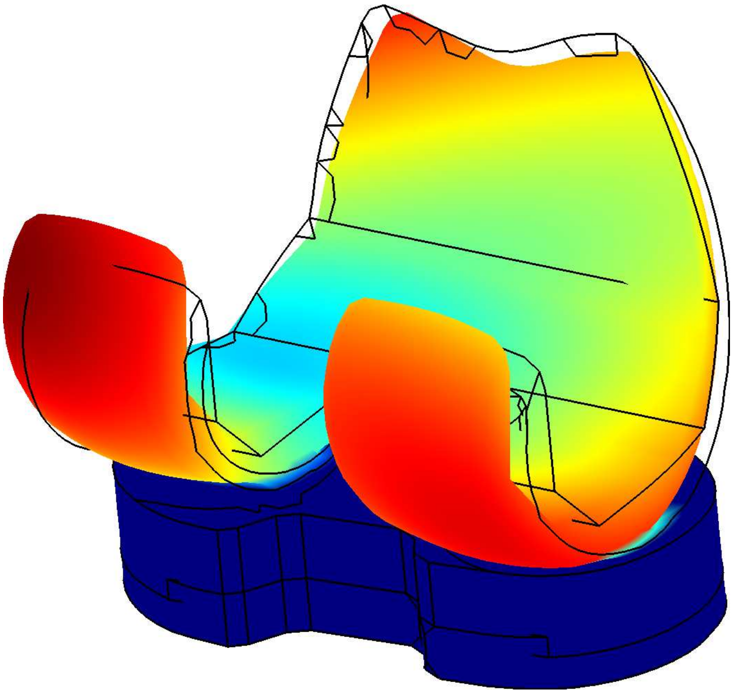


Figure 3: Total displacement

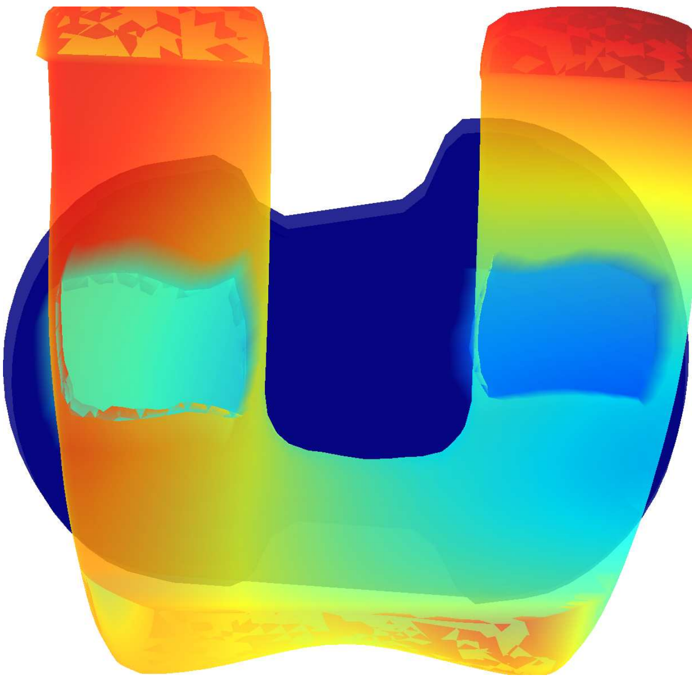


Figure 4: Total displacement - top view

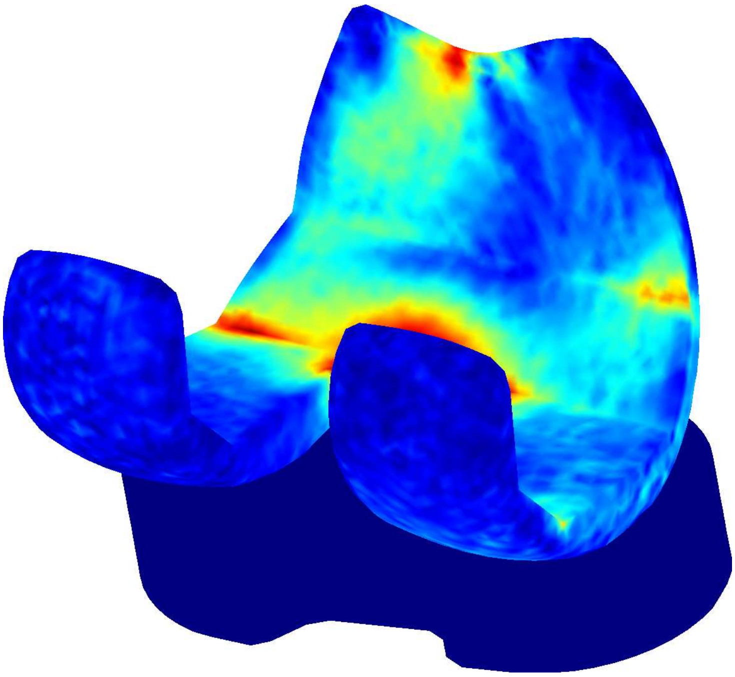


Figure 5: Von Mises stress

Acknowledgements

The author has been supported by the Ministry of Education, Youth and Sport of the Czech Republic, MSM4977751301.

References

[1] J. Daněk. *Use of MATLAB for domain decomposition method for contact problems in elasticity.* In 13th Annual Conference Proceedings of Technical Computing Prague 2005, Prague, Humusoft, 2005.

[2] J. Daněk. *Using Matlab for solving contact problem in elasticity.* In 11th Annual Conference Proceedings of Technical Computing Prague 2003, Prague, Humusoft, 2003.

[3] website `http://comsol.com`

[4] website `http://www.medin.eu/en/produkty/produkty.html?cat=3&productsID=77`

Ing. Josef Daněk, Ph.D.
Department of Mathematics
University of West Bohemia
Univerzitní 22
306 14 Pilsen
Czech Republic
E-mail: danek@kma.zcu.cz