EXPERIMENT-BASED FE SIMULATION OF ONCOLOGY KNEE ENDOPROSTHESIS

Lukas Zach*, Svatava Konvickova, Pavel Ruzicka

Czech Technical University in Prague,
Faculty of Mechanical Engineering,
Laboratory of Biomechanics, Prague, Czech Republic
* e-mail: Lukas.ZACH@fs.cvut.cz

Abstract

This paper presents two FE simulations of an oncology knee-joint endoprosthesis. Firstly dynamic experiment-based analysis traditionally made on special knee simulator has been calculated. Another FE simulation of an oncology knee-joint endoprosthesis was made on a complex model based on previously developed geometric model of a lower limb. The presented paper aims to prove the potential of finite element method in biomechanics, especially in development of joint endoprosthesis.

[Engineering of Biomaterials, 106-108, (2011), 10-12]

Introduction

Finite element method (FEM) has become a useful tool in classical mechanics while study of several mechanisms like fractures and defect origins and propagation. It can also become a valuable and effective tool for biomechanical devices development. Quite fast and easily modifiable models allow studying wide range of problems (static, dynamic) while using different boundary conditions (type of loading).

There are two groups of FEA of lower extremity models. The first ones are used to simulate a behavior of a healthy knee joint in-vivo [1-4] and the second group which deals with a knee joint after a total knee endoprosthesis (TKE) implantation [5,6].

Fractures and malfunctions of joint and bone implants has different causes. Generally, they can be sorted by two causes – biological and mechanical. Among the biological sources of implant damages, especially implant loosening and infection are well described in literature. In contrary, a stem fracture or a UHMWPE parts defect are typical causes of mechanical defects (FIG. 1).

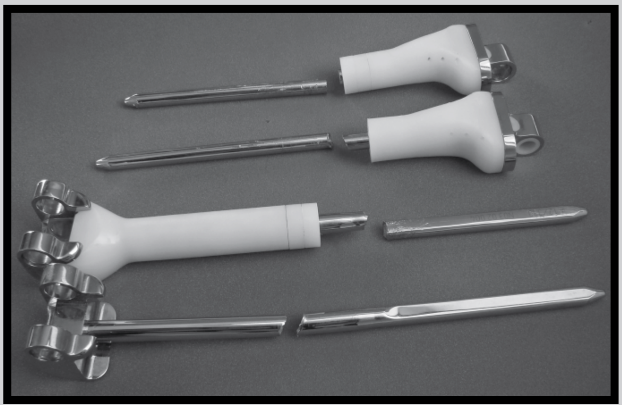

FIG. 1. Mechanical defects of oncology knee endoprosthesis.

To understand better the mechanical reasons for oncology knee implants destruction, the presented work has been made. For simulation of a loading by knee simulator, all boundary conditions are in accordance with ISO 14243-3: 2004 [6-8], where a manner of mechanical testing is defined. For real simulation of a behavior of a knee implant in human body, our own data and foundings cited in literature were applied.

Materials and methods

For presented nonlinear contact static analyses, solved in Abaqus CAE, an oncology knee endoprosthesis made by Prospon [9] has been chosen. The implant is made from titanium alloy Ti6Al4V. Tibial component is partly manufactured from UHMWPE - ultrahigh molecular weight polyethylene. The following simulations were made for two cases: UHMWPE bushings in case of knee simulator model and PEEK-OPTIMA® in case of whole limb model.

Following TABLE 1 summarize isotropic and homogenous material properties of all parts. Ideally plastic material model of UHMWPE is described in detail in FIG. 2.

TABLE 1. Material properties.

	Young modulus [MPa]	Max. tensile stress [MPa]	Poisson's ratio [-]
TiAl6V4	113800	900	0,34
PEEK	3650	90	0,44
UHMWPE	820	100	0,44

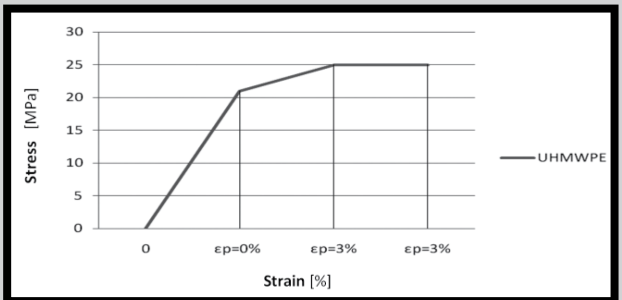

FIG. 2. Ideally plastic material model of UHMWPE.

As already mentioned, all boundary conditions in case of knee simulator (simplified knee model) are in accordance with ISO 14243-3: 2004 [8].

For real simulation of loading in stand (complex model featuring bone, muscle and ligament structures), boundary conditions defined by Vilimek [10] were applied. Totally 32 muscles, all three long bones together with patella defined the geometric model of the lower limb [11].

Results

Type of loading given by ISO 14243-1 [6-8] caused corresponding response in all parts of the knee oncology endoprosthesis. The most critical areas on the implant follow.

UHMWPE bushings which make parts of the hinge between the femoral component and tibial plateau are maybe the most critical plastic part of the endoprosthesis. Though the peak values between 16-28 MPa (FIG. 3) are reached for the bushings, it is only a case of limited number of elements so these results can be supposed to be "mesh errors". Nevertheless, these parts demand special attention and further study to eliminate an occurrence of PE wear.

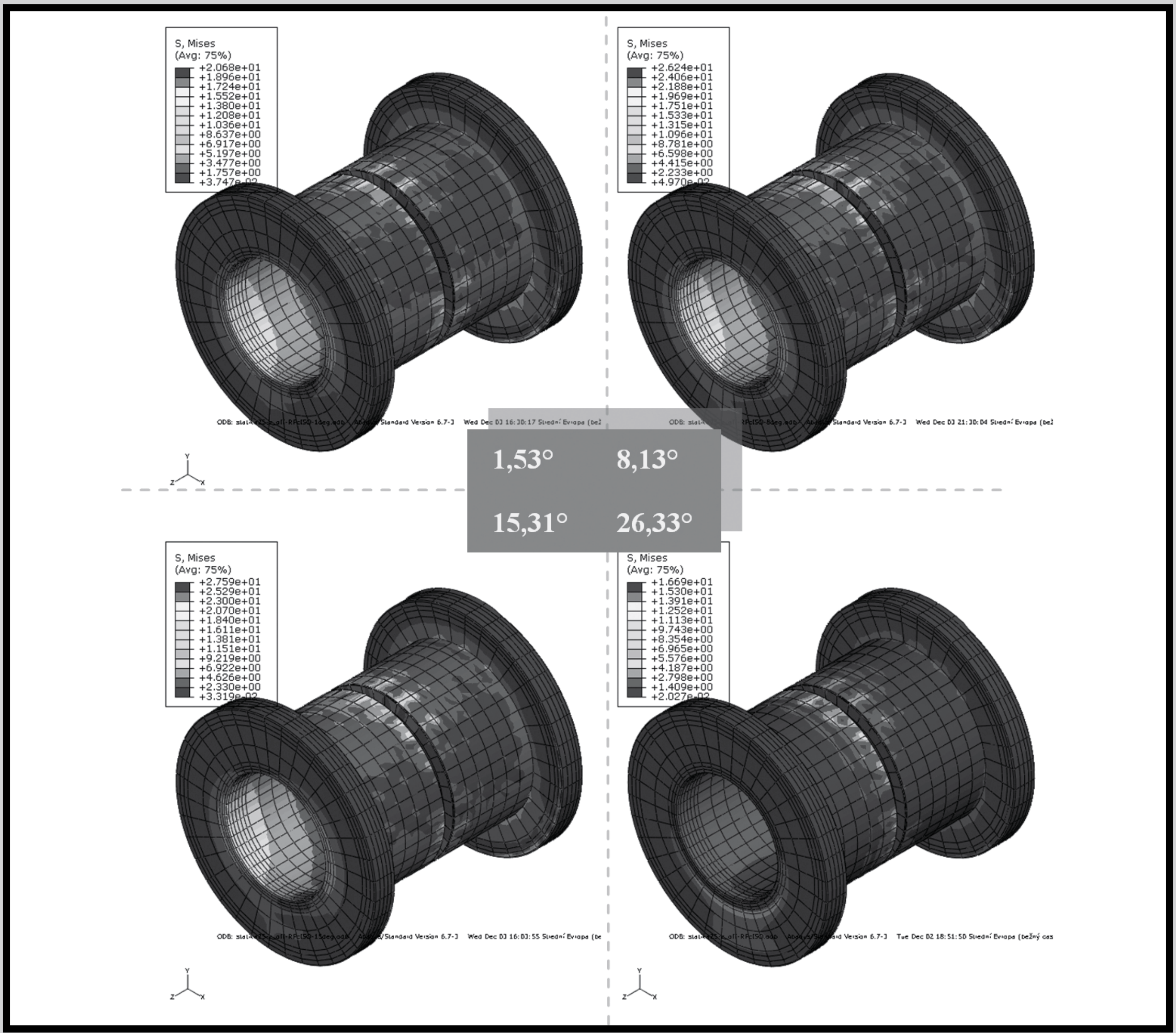

FIG. 3. Stress distribution (von Mieses theory) on surface of UHMWPE bushings [MPa].

As a part of the development the UHMWPE bushings were replaced by PEEK bushings which were used also in real simulation of the whole lower limb with implanted hinge-knee. The reached peak value of stresses according to von Mises theory were approx. 28 MPa (FIG. 4).

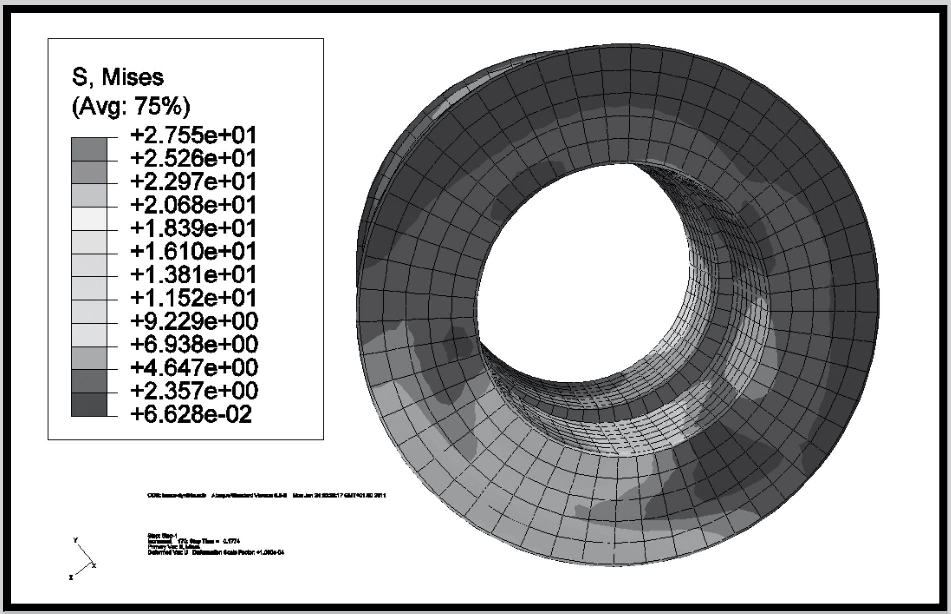

FIG. 4. Stress distribution (von Mieses theory) on surface of UHMWPE bushings in full extension [MPa].

Conclusions

One of the most common reasons of joint endoprosthesis malfunction leading to a reoperation is a mechanical defect or an implant loosening (or their combination). Appropriate design of the implant can dramatically eliminate this risk. As a useful tool for endoprosthesis development, a finite element method can be used, providing that the anatomical relation or mechanical test standards are kept. Based on our experiences with experimental and numerical simulations two presented FE models were analyzed.

A goal of this paper was, using an oncology knee implant, to make up a computer simulation of loading of TKE during the dynamic mechanical test defined by ISO 14243 [8] and loading of TKE in stand while profiting of formerly presented complex model of lower limb [11]. The results focused on the most critical areas of the endoprosthesis, i.e. UHMWPE bushings. The dynamic analyses and the static analysis of the complex model with the PEEK bushings pointed out the peak values of HMH stress of 28 MPa (FIG. 3-4). The analysis of PEEK-based model showed suitability of this type of the oncology knee. The results will be useful for our upcoming dynamic models and for further development of the oncology knee endoprosthesis. Since the oncology implants are produced as an individual replacement, finite element method represents a time and money saving method of the implant production.

Acknowledgements

This research was supported by Ministry of Education project Transdisciplinary Research in Biomedical Engineering II., No. MSM 6840770012 and by grant of Technology Agency of the Czech Republic TACR01010185.

References

[1] J.M.T. Penrose (2002) Development of an accurate three dimensional finite element knee model. Comp. Meth. in Biomech. and Biomed. Eng. 5, pp. 291-300.
[2] J.A. Heegaard (2001) A computer model to simulate patellar biomechanics following total kneee replacemnet: the effects of femoral component alignment. Clinical Biomech. 16, pp. 415-423.
[3] P. Beillas, et al (2004) A new method to investigate in vivo knee behavior using a finite element model of the lower limb. J Biomech. 37, pp. 1019-1030.
[4] A.C. Godest (2002) Simulation of a knee joint replacement during a gait cycle using explicit finite element analysis. J. Biomech. 35, pp. 267-275.
[5] J.P. Halloran (2004) Explicit finite element modeling of total knee replacement mechanics. J. Biomech. 38, pp. 323-331.
[6] ISO 14243-1. Implants for surgery - Wear of total knee-joint prostheses - Part 1: Loading and displacement parameters for wear-testing machines with load control and corresponding environmental conditions for test. 2002.
[7] ISO 14243-2. Implants for surgery - Wear of total knee-joint prostheses - Part 2: Methods of measurement. 2000.
[8] ISO 14243-3. Implants for surgery - Wear of total knee-joint prostheses - Part 3: Loading and displacement parameters for wear-testing machines with displacement control and corresponding environmental conditions for test. 2004.
[9] Prospon, s.r.o., Jiriho Voskovce 3206, Kladno, 272 01, Czech Republic. On-line at: http://www.prospon.cz/.
[10] M. Vilimek (2005), The challenges of musculotendon forces estimation in multiple muscle systems, [PhD Thesis]. Prague: Czech Technical University - Fac. of Mechanical Engineering, 108 p.
[11] L. Zach, S. Konvickova, P. Ruzicka (2008), Finite Element Analysis of Lower Limb. Engineering of Biomaterials, vol. 11, no. 77-80, p. 15-18, ISSN 1429-7248.