L. ZACH, L. KUNČICKÁ, P. RŮŽIČKA, B. BAGHIROVA

ISSN 0543-5846
METABK 55(3) 349-352 (2016)
UDC – UDK 616.314-089.28:517.083.5=111

EXPERIMENTAL AND NUMERICAL ANALYSIS OF A KNEE ENDOPROSTHESIS NUMERICAL MODEL

Received – Primljeno: 2015-08-31
Accepted – Prihvaćeno: 2015-12-26
Original Scientific Paper – Izvorni znanstveni rad

The aim of this study is to create and verify a numerical model for a Medin Modular orthopedic knee-joint implant by investigating contact pressure, its distribution and contact surfaces. An experiment using Fuji Prescale pressure sensitive films and a finite element analysis (FEA) using Abaqus software were carried out. The experimental data were evaluated using a special designed program and were compared with the results of the analysis. The designed evaluation program had been constructed on the basis of results obtained from a supplementary calibration experiment. The applicability of the numerical model for the real endoprosthesis behavior prediction was proven on the basis of their good correlation.

Key words: endoprosthesis; knee joint; numerical; analysis; FEM

INTRODUCTION

The first hinged type knee joint implant was made in 1954 from Co – Cr – Mo alloy, which still remains one of the most commonly used materials for fabrication of implants together with Ti – 6Al – 4V and Ti – Ni alloys [1-6]. Hinged implants are currently used in specific cases, such as for oncological implants, or for cases of severe damage [7]. In 1968, a polycentric implant model was developed and a new alloarthroplasty era started. During the following years, anatomic condylar prostheses with polyethylene menisci were designed.

The finite element method (FEM) has already been a standard development tool in engineering [8, 9]. The aim of various FE analyses in biomechanics is to advance joint implants development and to make it more effective. E.g., Halloran et al. [10] designed a FE model of a total knee replacement to be used for explicit dynamic analyses, and Guess et al. [11] created a FE knee model with implemented menisci suitable also for neuromusculoskeletal analyses.

This paper is focused on design and experimental verification of a numerical model of the Medin Modular (MM) orthopaedic knee-joint implant. The prosthesis has several variants (for primary, revision and stabilized replacements). Therefore, the designed model can be further used for analyses of individual cases. To verify the model, the contact pressure and its distribution and contact surfaces were investigated. The analysed parameters were selected based on researches published elsewhere, such as. ref. [10]. Experimental measurements were performed using the Fuji Prescale pressure sensitive films. The films are a simple solution to measure pressures and stresses in a wide research field range, from the aviation industry, through the food industry, to biomechanics. Subsequently, we evaluated the experimental data with the aid of our own Matlab script that we had programmed specially for the purposes of this experiment. Finally, we created a Finite Element numerical model of the prosthesis on the basis of the experimental data using a finite element analysis in the Abaqus software and compared it with the experimental results.

MATERIALS AND METHODS

Oncological prosthesis

Medin Orthopedics is the largest Czech Producer of medical implants (www.medin.eu). The femoral, as well as the tibial, components of the MM implant can be fixed to the bones using bone cement or screw plugs, according to the implantation type. The tibial component consists of an anchoring part with a stem made of Ti – 6Al – 4V alloy, which respects different shapes of the medial and lateral tibia condyle, and an UHMWPE tibial insert. The components are fixed to the bones using screw plugs.

Fuji Prescale technology

Measurement of the contact pressure and its distribution is performed by the very thin Fuji Prescale films (www.fujifilm.com). These films are produced in two types, mono-sheet and two-sheet, and seven variants which cover different ranges of measurable pressures.

L. Zach, P. Růžička, Czech Technical University in Prague, Faculty of Mechanical Engineering, Prague, Czech Republic
L. Kunčická, B. Baghirova, VŠB-Technical University of Ostrava, Faculty of Metallurgy and Materials Engineering, Ostrava-Poruba, Czech Republic

Construction of evaluation curves

To check the properties of the pressure sensitive films and to construct the calibration and evaluation curves, supplemental calibration experiment was performed, during which the device with a film was loaded with maximum pressure in the range from 2,5 MPa to 50 MPa. For the experiment, two types of Fuji films were used; LW (2,5 – 10 MPa pressure range, two-sheet type) and MS (10 – 50 MPa pressure range, mono-sheet type). The experiment was performed using the face of a 20 mm diameter and 80 mm long steel cylinder with 1 mm / min loading feed and 5 s load hold time. The Fuji film was put on a fixed steel table of 120 mm diameter with shoulders to be connected to the MTS 585 Mini Bionix testing device.

After measurements, the film samples were scanned with 1 200 dpi resolution. The scanned film samples were imported to the Matlab software in the RGB color scale, which enabled to work with three matrices describing the red, green and blue color intensities. Subsequently, the data were modified with the aid of the Matlab script; the red color intensity outside the red patch was set to be 0, therefore the red color intensity inside the red patch could be measured in the scale from 0 to 255. Finally, the average red color intensity values were calculated in each of the red patches.

Contact pressure values were calculated from the formula:

$$p = \frac{F}{S} \tag{1}$$

where p is the contact pressure / MPa, F is the experimental loading force / N and S is the surface area of the cylinder face / mm^2. The final evaluation curves were obtained by a second degree polynomial curve fit in the red color intensity – computed contact pressure dependence. The curves for both the film types are depicted in Figure 1.

Based on the results of the measurements and computations, a special Matlab application for evaluation of the experimental data was created.

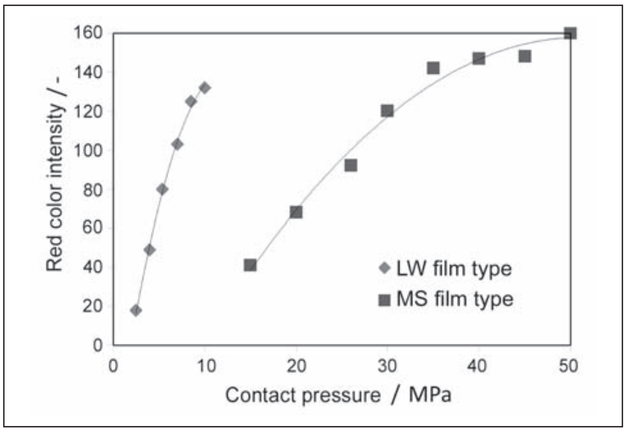

Figure 1 Evaluation curves for the pressure sensitive films were constructed on the basis of our own calibration experiment

Contact pressure measurements

The measurement was performed on a size 68 Medin Modular endoprosthesis, right side design. The tibial component was, for the purposes of the experiment, represented only by a polyethylene plateau. Both the components were gripped in a special device during testing.

The films were cut into 3,2 x 4,2 cm large pieces for both the condyles. For proper determination of the position of the contact area, the films were equipped with guidelines in the centers of their sides. The films were fixed to the femoral and tibial contact surfaces with double-sided adhesive tapes.

The experiment was performed at 23 °C temperature and 23 % relative humidity on an MTS 585 Mini Bonix testing device with the loading feed of 1 mm / min and 5 s load hold time. The measurements were carried out in 0 ° knee joint flexion for loading forces from 500 N to 3 000 N with a step of 500 N.

Finite Element Analysis (FEA)

The FEA was based on our own 3D CAD models and respected the basic conditions of the experiment. The FE mesh was semiautomatic generated and consisted of C3D10M, second-order 10-nodes modified tetrahedral elements with hourglass control associated with their internal degrees of freedom. A Node-to-Surface contact was used between the tibial and femoral endoprosthesis components. A Hard Contact, i.e. contact with no possibility of penetration of the bodies was applied in the normal direction. A Tie connection was defined between the femoral component and the fixing device (a fixed connection of the nodes of the contact surfaces).

The friction coefficient value was set to be 0,2 [12]. The fixation device was made of X5CrNi18-10 stainless steel. Therefore, the material was defined to be homogenous and isotropic with elastic behavior. The same material model was used for the Co – Cr – Mo femoral component. Due to possibility of occurrence of plastic deformation the UHWPE behavior was described by an

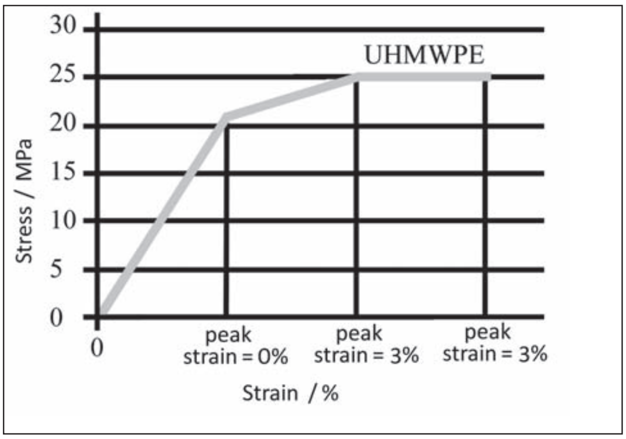

Figure 2 The stress–strain dependence of the UHMWPE elastic–plastic material model

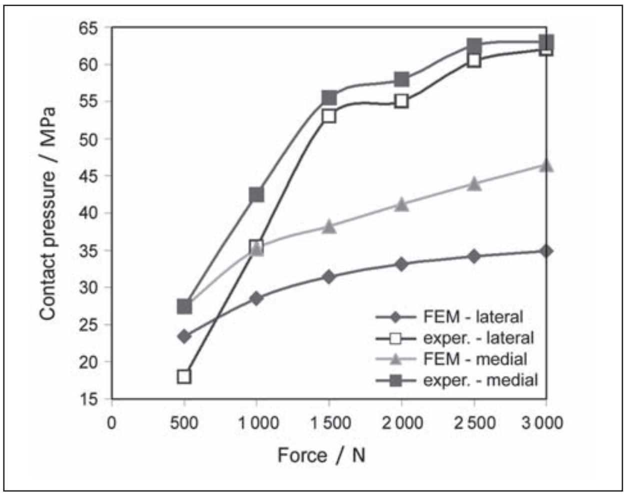

Figure 3 Dependence of experimental and predicted contact pressures for both the condyles

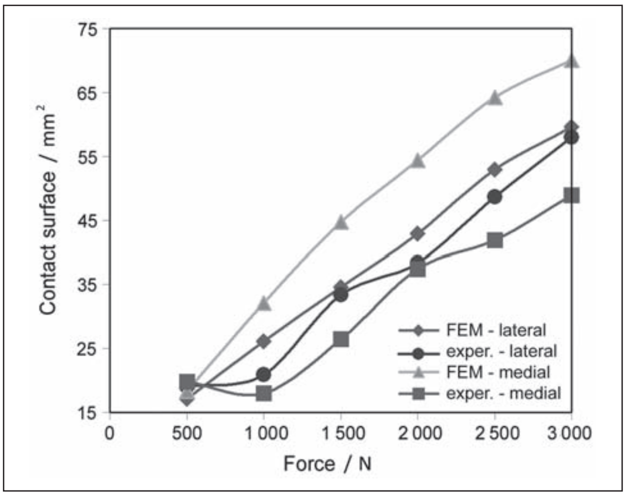

Figure 4 Dependence of experimental and predicted contact surfaces for both the condyles

elastic-plastic model with the stress-strain dependence depicted in Figure 2.

RESULTS AND DISCUSSION

Contact pressures

The maximum contact pressure calculated from FE model increased with increased loading force up to the value of 46,47 MPa for the medial condyle. The maximum contact pressure values obtained during the experiment and during the FEA for the lateral and medial condyles are summarized in Figure 3.

The experiment was carried out twice for each of the loading forces and the average maximum contact pressure values were calculated. The differences between simulation and experimental values are smaller for medial condyle than for the lateral one. There is a maximum deviation of 43,7 % on the lateral condyle for the 3 000 N loading force. The minimum deviation is 0,36 % on the medial condyle for the 500 N force.

Contact surfaces

The dependence of the contact surface values resulting from the experiment and the FE analysis for both the condyles is depicted in Figure 4. In the dependence, conformity of the experimental and the simulated contact surface values for the lateral condyle is obvious.

The deviation range for the lateral condyle computations was from 3,5 % for a 1 500 N loading force to 19,9 % for 1 000 N loading force which seems to be acceptable. For the medial condyle an unexpected decrease of measured contact surface occurred for the loading force of 1 000 N causing deviation of 43,9 % from the simulation data. As far as the rest of medial condyle experimental data also differed significantly from the simulation results it should be stated that for the medial condyle contact surfaces measurement the verification is not convincing.

CONCLUSIONS

The aim was to create a FE model of a Medin Modular knee endoprosthesis and to validate the model by an experimental analysis. The experimental data evaluation was performed using own evaluation curves created with the aid of own Matlab computer program on the basis of test measurements. The results showed a good correlation of the experimental and the predicted data, especially as regards the lateral condyle contact surfaces. On the basis of the results, the final finite element model can be considered to be conformant to the real endoprosthesis behavior and thus applicable for the creation of more complex individual assemblies with ligaments and muscles.

Acknowledgements

This paper was created within the Project no. LO1203 "Regional Materials Science and Technology Centre – Feasibility Program" funded by Ministry of Education, Youth and Sports of the Czech Republic and under the grant of Ministry of Industry of the Czech Republic FR-TI3/221.

REFERENCES

[1] J. Nedoma, J. Stehlík, I. Hlaváček, J. Daněk, T. Dostálová, P. Přečková, Total Replacement of Human Joints, Mathematical and Computational Methods in Biomechanics of Human Skeletal Systems, New Jersey, 2011, pp. 34-73.

[2] L. Kuncicka, R. Kocich, J. Drapala, V. A. Andreyachshenko, FEM simulations and comparison of the ECAP and ECAP-PBP influence on Ti6Al4V alloy's deformation behaviour, Proceedings of the 22nd International conference on metallurgy and materials, Brno, 2013, Czech Republic, pp 391-2396

[3] R. Kocich, I. Szurman, M. Kursa, J. Fiala, Investigation of influence of preparation and heat treatment on deformation behaviour of the alloy NiTi after ECAE, Mater. Sci. Eng. A 512 (2009) 100-104.

[4] R. Kocich, M. Kursa, I. Szurman, A. Dlouhy, The influence of imposed strain on the development of microstructure and transformation characteristics of Ni-Ti shape memory alloys, J. Alloy. Compd. 509 (2011) 2716-2722.

[5] L. Zach, L. Kuncicka, P. Ruzicka, R. Kocich, Design, analysis and verification of a knee joint oncological prosthesis finite element model, Comput. Biol. Med. 54 (2014) 53-60.

[6] I. Szurman, R. Kocich, M. Kursa, Using of ultrasonic methods for determination of the elastic moduli on the TiNi based alloys, Metalurgija 53 (2014) 357-360.

[7] F. F. Buechel, M. J. Pappas, The New Jersey LCS knee replacement system biomechanical rationale and comparison of cemented and noncemented results, Contemp. Orthopaedics 14 (1987) 52-58.

[8] R. Kocich, A. Machackova, V.A. Andreyachshenko, A study of plastic deformation behaviour of Ti alloy during equal channel angular pressing with partial back pressure, Comp. Mat. Sci. 101 (2015) 233-241

[9] R. Kocich, M. Greger, A. Machackova, Proceedings of the 19nd International conference on metallurgy and materials, Roznov Pod Radhostem, 2010, Czech Republic, pp 166-171.

[10] J. P. Halloran, A. J. Petrella, P. J. Rullkoetter, Explicit finite element modeling of total knee replacement mechanics, J. Biomech. 38 (2005) 323-331.

[11] T. M. Guess, G. Thiagarajan, M. Kia, M. Mishra, A subject specific multibody model of the knee with menisci, Med. Eng. Phys. 32 (2010) 505-515.

[12] M. Vilimek, Musculotendon Forces Derived by Different Muscle Models, Acta Bioeng. Biomech. 2 (2007) 41-47.

Note: The paper was translated by Lector of TU Ostrava