L. ZACH, L. KUNČICKÁ, A. MACHÁČKOVÁ, P. RŮŽIČKA, M. POHLUDKA, J. JUŘICA

ISSN 0543-5846
METABK 54(4) 691-694 (2015)
UDC – UDK 669.295:620.73:519.6=111

FINITE ELEMENT ANALYSIS OF Ti-BASED KNEE-JOINT IMPLANT

Received – Primljeno: 2014-07-10
Accepted – Prihvaćeno: 2015-01-30
Preliminary Note – Prethodno priopćenje

The focus of this paper was on finite element analysis of a PROSPON oncological knee endoprosthesis. The 3D CAD knee joint model, the designed FE PROSPON prosthesis model into which was integrated, was created on the basis of Visible Human Project CT scans. Analyses of stress state and contact pressures were performed in the knee-bending position within 15,4° - 69,4° hip joint flection range. The results showed that the maximum achieved stress did not exceed the yield strength (90 MPa) of the material. The results of the stress state were in accordance with the distribution of contact pressure.

Key words: Ti-alloys, knee joint, implant, Finite Element Method (FEM)

INTRODUCTION

Over the past decades, the Finite Element Method (FEM) has emerged to provide general applicability and necessary accuracy to become widespread for various stress analyses. Using FEM, the body is divided into a number of subdomains called *elements*. Since element size and shape and approximating scheme can be varied according to the given problem, the method can accurately simulate solutions of complex geometry problems.

Computer aided analyses have already been a standard development tool, e.g. in mechanical [1], thermal [2] or materials [3] engineering.

As severe plastic deformation (SPD) methods development and use is ever increasing [4], FE analyses are used to simulate conditions during these processes. Among the most widely analysed SPD methods are ECAP [5,6] and its modifications – ECAP with non-equal channels (NECAP) [7], with partial back pressure (ECAP-PBP) [8], with twist extrusion (TCAP) [9], with twist extrusion and multiple bendings (TCMAP) [10], ECAP-CONFORM [11] or high pressure torsion (HPT) [12].

The analyses are primarily carried out to predict deformation behaviour of a material under certain thermal and friction conditions.

The simulations are useful especially for materials with lower formability, such as these with high content of Ti [13], intermetallic alloys [14] or Mg [15].

In biomechanics, e.g. during the process of joint implants development, where experimental in vivo measurements can only be performed in certain conditions, FEM is a suitable time and money saving tool. Validation of a model can be performed in different ways, e.g using a knee simulator [16,17], or experimentally [18]. The simulations are mostly performed to investigate the contact pressure and its distribution [19], overall stiffness [20], or static and dynamic implant and/or joint behaviour [21].

Another advantageous use of computer simulations is to predict behaviour of a newly developed and designed products and devices, such as prediction of heat conduction in a microchannel heat exchanger [22], and characterisation of a new Water Jet Driven Particle Collider device [23].

The aim of this paper was to perform a FEM stress-strain analysis of the PEEK-OPTIMA® hinge pin bushing, the most loaded part of the PROSPON oncological knee endoprosthesis. Analysis of contact pressures was performed as well. The results were compared to the yield strength of the material.

THEORY AND METHODOLOGY

Oncological implant

The PROSPON oncological implants are made to order individually, the used oncological implant consisted of femoral component, femoral stem, rotating hinge post, tibial plateau, tibial base plate, tibial stem, medial/lateral hinge pin bushing, hinge pin, and hinge pin plug.

3D model

The construction of the 3D model was performed using CT scans acquired within the (http://www.nlm.nih.gov/research/visible/visible_human.html) Visible Human Project. After acquisition, the CT scans were adjusted and segmented and the 3D reconstruction pro-

L. Kunčická, A. Macháčková, M. Pohludka, J. Juřica - VŠB – Technical University of Ostrava, Faculty of Metallurgy and Materials Engineering, Czech Republic
L. Zach, P. Růžička, Czech Technical University in Prague, Faculty of Mechanical Engineering, Czech Republic

cess, during which the final lower extremity CAD model was created, followed.

According to White [24], 25 main muscles were implemented to carry out a realistic 3D FEM analysis. 12 ligaments were implemented in the model in the same way as the muscles [25,26]. All the muscles were replaced by straight lines connecting the points of attachments of muscles to bones, considered as gravity centres of bone surfaces [24]. The final complete assembly was composed of bones (femur, fibula, tibia and patella), muscles and ligaments according to the above-mentioned model description.

Finite element assembly

The analysis was performed using the ABAQUS software. The FE mesh of the assembly was a combination of several types of elements, the overall number was 338 003. CONN3D2 connector elements of the AXIAL type with two nodes and one degree of freedom were used for the muscles. In the analysis, only the patellar tendon, represented by another connector of the AXIAL type (rigidity of 1 000 N/m), was specified. All the other tendons were components of the general lower extremity model. The ankle was replaced by a combination of JOIN and CARDAN connectors, the foot was replaced by a rigid part consisting of five points. Fixation of the implant to the bones was replaced by a solid TIE connection. A bond of the same type was used for the connection between the femur and tibia bones and between all the PROSPON endoprosthesis components. For the following contact pairs, surface-to-surface contacts were used: femur – patella, femoral component – patella, femoral component – hinge pin bushing, hinge pin – hinge pin bushing. The friction factor value for contact pairs was 0,2 or 0,5 for pairs connecting to a bone [26]. Boundary conditions were taken mainly from Vilímek's work (musculotendons' forces) [26]. The analysed part is schematically depicted in Figure 1.

A simplified constitutive model was used for the materials, homogenous, isotropic or linear material behaviour was defined. For all the materials except the UHMWPE, an elastic material model was used. This material was defined using an elastic-plastic material model due to the possibility of plastic deformation development.

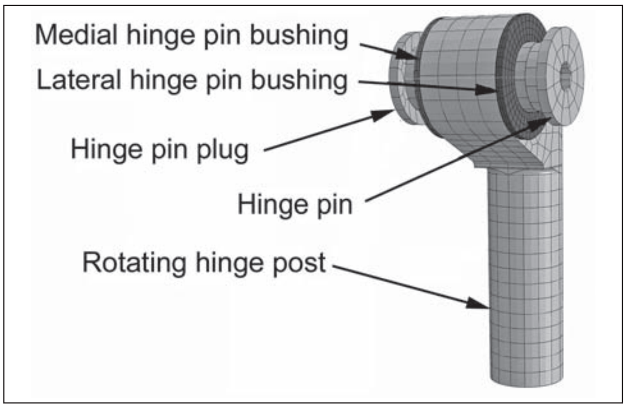


Figure 1 Detail of the hinge pin bushing in the assembly

The whole assembly was solved as a dynamic implicit model with 100 automatic steps (from 15,4° to 69,4° hip joint flection).

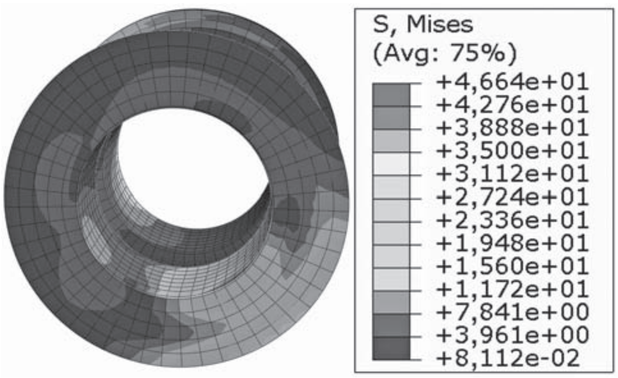
RESULTS

Von Mises stress

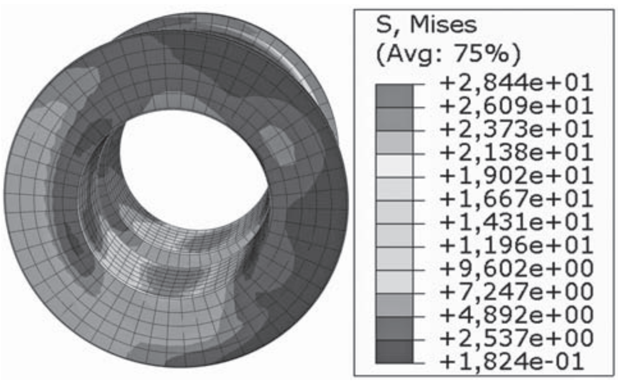
The results of the Von Mises stress analysis are to be seen in Figure 2a (the initial position) and Figure 2b (the final position). According to the results, the max. stress value of 46,64 MPa was achieved in the initial flection angle. At low hip joint flection angle, the maximum stress value effects on the lateral bushing inner contact surface, whereas the maximum value decreases as the flection angle increases. At the same time, the medial bushing side becomes affected by the maximum stress value as well.

Contact pressure

In Figures 3a (initial) and Figure 3b (final) the contact pressure distribution for both the positions are shown. The maximum contact pressure value of 60,32 MPa is seen in Figure 3a. In the initial position, the maximum contact pressure takes effect only on the lateral bushing inner contact surface. As the extremity goes to the knee-bending position, the maximum con-

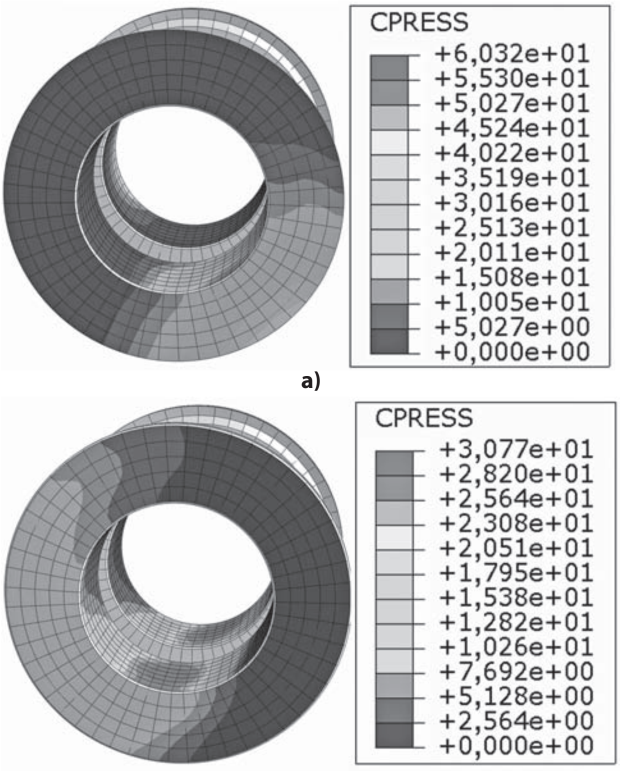


a)



b)

Figure 2 Von Mises stress on PEEK bushing surfaces (HMH hypothesis) a) 15,4°; b) 69,4° hip joint flection



a)

b)

Figure 3 Contact pressure distributions on PEEK bushing surfaces a) 15,4°; b) 69,4° hip joint flection

tact pressure value decreases to 30,77 MPa and the distribution of the pressure becomes more homogenous on both the sides of the bushing.

DISCUSSION

From the Von Mises stress analysis is evident that the maximum stress value (46,64 MPa) did not exceed the yield strength (90 MPa) of the PEEK material Therefore, the condition of plastic stability was met.

Analysis of contact pressure was carried out in order to confirm the stress state investigated during the Von Mises stress analysis. The contact pressure distribution was in accordance with the distribution of Von Mises stress. As can be seen in Figures 3a and 3b, the maximal pressure occurred locally on the side of the lateral bushing in the lower hip joint flection angle. As the hip joint flection angle increased, the distribution of contact pressure became more homogenous and its maximal value decreased.

The results also indicate that the effect of the knee-bending position on the endoprosthesis is not so harmful since the stress distribution in the bushing in the larger flection was more homogeneous than in the lower flection angle.

However, pressure reduction in the deeper knee-bending position can be due to loads being transferred through the other bony structures or ligaments of the general lower extremity model. A different number of ligaments, muscles and bony structures, can exist in individual cases.

Moreover, the vector of the loading force changes with increasing flection angle.

CONCLUSION

An analysis focused on the PROSPON implant PEEK-OPTIMA® hinge pin bushing was carried out in order to investigate the plastic stability of the material. The designed implant model was implemented into an assembly of lower extremity bones, muscles and ligaments. The analysis was carried out in the knee-bending position within the 15,4° - 69,4° hip joint flection range. The condition of plastic stability was met; the maximum Von Mises stress (46,6 MPa) did not exceed the yield strength of the material (90 MPa). Contact pressure distribution confirmed the Von Mises stress analysis results.

Acknowledgements

This paper was elaborated within the following projects: SP2014/100 “Investigation of deformation behaviour of materials using deformation simulator and laboratory rolling”, SP2014/62 “Specific research in metallurgical, materials and process engineering” and under the grant of Ministry of Industry of the Czech Republic FR-TI3/221.

REFERENCES

[1] L. M. Hlaváč, B. Strnadel, J. Kaličinský, L. Gembalová, The model of product distortion in AWJ cutting, The International Journal of Advanced Manufacturing Technology 62 (2012) 157-166.

[2] R. Kocich, J. Kliber, M. Herout, Finite elements method (FEM) simulation based prediction of deformation and temperature at rolling of tubes on a pilgrim mill, Metalurgija 48 (2009) 267-271.

[3] G. A. Quadir, I. A. Badruddin, N. J. Salman Ahmed, Numerical investigation of the performance of a triple concentric pipe heat exchanger, International Journal of Heat and Mass Transfer, 75 (2014) 165-172.

[4] Y. T. Zhu, T. C. Lowe, T. G. Langdon, Performance and applications of nanostructured materials produced by severe plastic deformation, Scripta Materialia, 51 (2004) 825-830. DOI: 10.1016/j.scriptamat.2004.05.006.

[5] R. Kocich, M. Greger, A. Macháčková, Finite element investigation of influence of selected factors on ECAP process, Metal 2010: 19th international metallurgical and materials conference, 2010, pp. 166-171.

[6] Y. T. Zhu, T. C. Lowe, Observations and issues on mechanisms of grain refinement during ECAP process, Materials Science and Engineering A, 291, (2000) 46-53.

[7] C. F. Gu, L. S. Tóth, Texture development and grain refinement in non-equal-channel angular-pressed Al, Scripta Materialia 67 (2012) 33-36.

[8] A. B. Naizabekov, V. A. Andreyachshenko, R. Kocich, Study of deformation behavior, structure and mechanical properties of the AlSiMnFe alloy during ECAP-PBP Micron 44 (2013) 210-217.

[9] R. Kocich, L. Kunčická, M. Mihola, K. Skotnicová, Numerical and experimental analysis of twist channel angular

pressing (TCAP) as a SPD process, Materials Science and Engineering A, 563 (2013) 86-94.

[10] R. Kocich, A. Macháčková, L. Kunčická, Twist channel multi-angular pressing (TCMAP) as a new SPD process: Numerical and experimental study, Materials Science and Engineering A, 612 (2014) 445-455. DOI: 10.1016/j.msea.2014.06.079

[11] G. I. Raab, F. F. Safin, T. C. Lowe, Y. T. Zhu, R. Z. Valiev, Ultrafine Grained Materials IV, 2006, 171-175.

[12] Y. P. Song, W. K. Wang, D. S. Gao, E. Y. Yoon, D. J. Lee, C. S. Lee, H. S. Kim, Hardness and microstructure of interstitial free steels in the early stage of high-pressure torsion, Journal of Materials Science 48 (2013) 4698-4704.

[13] R. Kocich, I. Szurman, M. Kursa, Shape Memory Alloys - Processing, Characterization and Applications, F. M. B. Fernandes (Ed.), 2013, pp. 27-52.

[14] M. Pohludka, J. Malcharczikova, V. Michenka, M. Kursa, T. Čegan, Metal 2013: 22nd international metallurgical and materials conference, 2013, pp. 1567-1572.

[15] R. Kocich, M. Kursa, A. Macháčková, Acta Physica Polonica A, 122 (2012) 581-587.

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

[17] T. M. Guess, G. Thiagarajan, M. Kia, M. Mishra, A subject specific multibody model of the knee with menisci, Medical Engineering & Physics 32 (2010) 505-515.

[18] C. Öhman, D. M. Espino, T. Heinmann, M. Baleani, H. Delingette, M. Viceconti, Visual Computing 27 (2011) 153-159.

[19] L. Zach, L. Kunčická. P. Růžička, R. Kocich, Design, analysis and verification of a knee joint oncological prosthesis finite element model, Computers in Biology and Medicine 54 (2014) 53-60.

[20] U. Simon, P. Augat, A. Ignatius, L. Claes, Influence of the stiffness of bone defect implants on the mechanical conditions at the interface–a finite element analysis with contact, Journal of Biomechanics 36 (2003) 1079-1086.

[21] M. F. Moran, S. Bhimji, J. Racanelli, S. J. Piazza, Computational assessment of constraint in total knee replacement, Journal of Biomechanics 41 (2008) 2013-2020. DOI: 10.1016/j.jbiomech.2008.03.020.

[22] M. I. Hasan, H. M. Hasan, G. A. Abid, Study of the axial heat conduction in parallel flow microchannel heat exchanger, Journal of King Saud University-Engineering Sciences 26 (2014) 122-131.

[23] L. M. Hlaváč, I. M. Hlaváčová, J. Vašek, P. Jandačka, J. Zegzulka, J. Wiliamsova, V. Mádr, R. Uhlář, Investigation Of Samples From the High-Velocity Water Jet Driven Micro/Nano Particle Collider, Pressure Vessels and Piping Conference of the American Society of Mechanical Eng., 2010, 119-126.

[24] S. C White, H. J. Yack, D. A. Winter, A three-dimensional musculoskeletal model for gait analysis. Anatomical variability estimates, Journal of Biomechanics 22 (1989) 885-893. DOI: 10.1016/0021-9290(89)90072-9.

[25] R. Crowninshield, M. H. Pope, R. J Johnson, An analytical model of the knee, Journal of Biomechanics, 9 (1976) 397-405. DOI: 10.1016/0021-9290(76)90117-2.

[26] M. Cadova, M. Vilimek, The necessity of physiological muscle parameters for computing the muscle forces: application to lower extremity loading during pedalling, Acta of Bioengineering and Biomechanics 11 (2009) 59-64.

Note: The paper by was translated by Zdeňka Kunčická, Metal Translations, Ostrava, Czech Republic