Available online at www.sciencedirect.com
ScienceDirect
Procedia Engineering 149 (2016) 257 – 262
Procedia Engineering
www.elsevier.com/locate/procedia

International Conference on Manufacturing Engineering and Materials, ICMEM 2016, 6-10 June 2016, Nový Smokovec, Slovakia

Finite element modeling of hip implant static loading

Katarina Colic[a]*, Aleksandar Sedmak[b], Aleksandar Grbovic[b], Uros Tatic[a], Simon Sedmak[a], Branislav Djordjevic[a]

[a]*University of Belgrade, Innovation Centre of Faculty of Mechanical Engineering, 11000 Belgrade, Serbia*
[b]*Faculty of Mechanical Engineering, University of Belgrade, 11000 Belgrade, Serbia*

**kbojic@mas.bg.ac.rs*

Abstract

In this paper a numerical investigation of replacement implant for partial hip arthroplasty is presented. The long-term stability of hip implants depends, among other things, on the loads acting across the joint. Forces occurring in vivo can be much greater than the recommended test values, because a typical gait cycle generates forces up to 6–7 times the body weight in the hip joint. A finite element analysis (FEA) was performed using 3-dimensional models to examine the mechanical behaviour of the femoral component at forces ranging from 2.5 to 6.3 kN. This implant design was chosen for numerical analysis because stress concentration in femoral component lead to implant fracture. Results show that the force magnitudes acting on the implant are of interest, and that they can cause implant stress field changes and implant stability problems, which can lead to implant failure.

Peer-review under responsibility of the organizing committee of ICMEM 2016

Keywords: hip implant failure, Ti-6Al-4V alloy, Finite element analysis, stress concentration, boimedical application design

1. Introduction

Total or partial hip replacement is a surgical procedure in which parts of the hip joint are removed and replaced with artificial parts, known as the prosthesis. [1, 2] Currently, titanium-based alloys, especially Ti-6Al-4V & Ti-6Al-7Nb, are the most commonly used materials for joint prostheses, being registered in ASTM standard as biomaterials.

While it is not possible to avoid failure, recent work has focused on predictive and design tools to enable more accurate prediction to avoid catastrophic failure of an implant. [3-9]. The goal of many studies that have been conducted using numerical methods in the field of total or partial hip arthroplasty has been to improve the overall reliability of orthopedic implants. [10-19]

During surgery and general handling of the prosthesis, scratches will inevitably appear on its surface, which will result in an intensification of the stress at those points and provide a location for crack growth propagation. Figure 1 shows the devastation that can occur to an improperly designed artificial hip implant. [3, 20]

In order to ensure prosthetic design safety relative to its mechanical behaviour, detailed analyses with different load cases need to be performed. In literature, static FEM analyses are typically performed using loads with a magnitude corresponding to body weight. [11,21,22] However, the effects of weight and sudden movement can increase the load to which the prosthesis is subjected by up to 10-20%, and in some cases even more significantly and this must be taken into account when estimating whether the prosthetic will fracture or fail due to fatigue. In order to investigate the difference in results predicted by standard tests of implants and real loads that can occur in practice, it is necessary to analyse the prosthesis under static loads corresponding to the body weight, as well as under maximum real load that is expected to occur during the walking cycle.

* Corresponding author. Tel.: +0-000-000-0000 ; fax: +0-000-000-0000.
E-mail address: author@institute.xxx

Peer-review under responsibility of the organizing committee of ICMEM 2016
doi:10.1016/j.proeng.2016.06.664

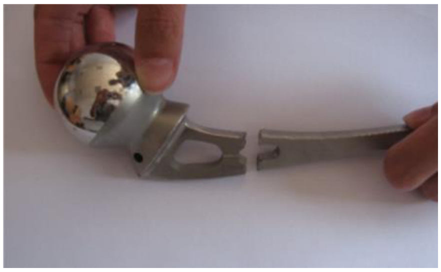

Figure 1. Artificial hip prosthesis failure

Performed in this paper is the stress analysis on a hip prosthesis during various patient activities, such as slow walking, climbing up and down stairs, as well as in the extreme case of tripping. These loads are calculated according to the experimental observations performed by Bergman *et al.* on a patient who weighs 860 N, within an age group of around 25 years. In these cases, it was possible to assume that the stage during which the patient leans on only one leg, defined during the walking cycle analysis, will last sufficiently long to enable approximation of problems using static calculations. Static load selected for the numerical analysis represents a person weighing 90 kg, and the loads given in Table 1 were assumed in accordance with it.

Table 1. Loading of hip joint for different problems

Activity	Maximum load (% of body weight)	Maximum force in joint (N)
Problem involving slow walking on a flat surface	282	2490
Problem involving climbing upstairs	356	3143
Problem involving tripping	720	6358
Problem involving climbing downstairs	387	3417

2. Finite element modelling and calculation

Model was set up with adequate boundary conditions, including fixing of the implant bottom surface along all degrees of freedom, and the load was applied in the appropriate direction relative to the top of the femoral head of the prosthesis.

FEM analysis of the prosthesis was performed using ABAQUS (Dassault Systèmes) software, for simulating slow walking on a flat surface for a model made of Ti-6Al-4V alloy. The most realistic models are considered those based on plasticity or viscoplasticity, in particular the HISS models, as they include other plasticity models as special cases. However, for evaluation of failure or ultimate loads the classical plasticity models are often used. In this part of the numerical investigation, three-dimensional static idealizations are considered, and constitutive models were used for elastic-plastic behaviour of biomaterial. Elastic-plastic von Mises conventional material model was used in this numerical analysis. For preliminary studies of mechanical behaviour of hip implant presented in this paper, linear elastic behaviour of a material was analysed, as a function of two elastic constants - Young’s modulus 120 GPa and Poisson’s ratio 0.3.

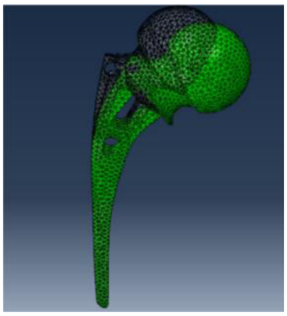

Figure 2. Comparative representation of the initial state and maximum load state of the implant

Analyses of three-dimensional stress states determined according to Von Misses criteria were performed, along with three-dimensional displacement fields depending on the loads of the implant. Shown in Figure 2 is a comparative graphical representation of implant displacement at the end of the calculation, under the effect of maximum load of 6 kN versus the initial non-deformed state.

Von Misses stresses on the stems, caused by the static analysis, are shown in Figures 3 and 4. Stresses were calculated in order to estimate the probability of prosthesis failure under the effect of maximum loads which can occur during a walking cycle. Results of numerical analyses of suggested models are shown in table 2.

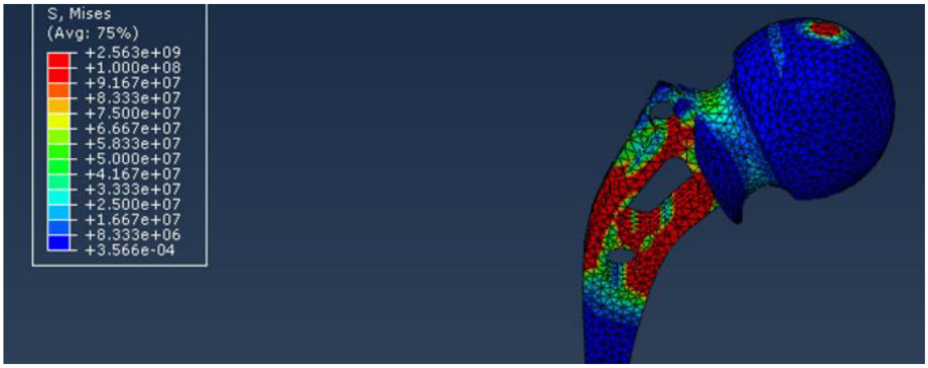

Figure 3. Von Misses stresses in the stem

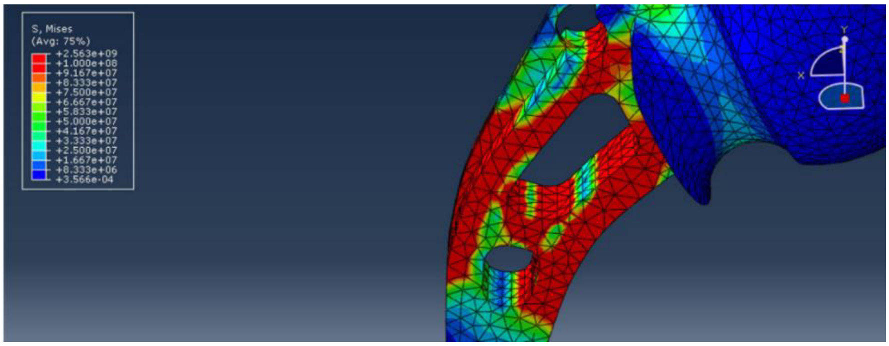

Figure 4. Von Mises stresses in the stem (detailed)

Table 2. Results of numerical analyses of suggested models

Problem	Maximum Von Mises stress (MPa)
Problem involving slow walking on a flat surface	256.3
Problem involving climbing upstairs	361.2
Problem involving tripping	535.5
Problem involving climbing downstairs	312.6

3. Discussion

Calculated Von Misses stresses, shown in Table 2, are significantly lower than yield stresses of Ti-6Al-4V (860 MPa). Maximum stress values were obtained in locations where stress concentrations were expected, as shown on selected relevant cross-sections. For the cross-section shown in Figure 5, the stress state in the implant cross-section with the hole corresponds to the location of the hole in the actual prosthesis, and it represents the location of critical stress concentration. It is also the exact area in which fracture occurred in the specimens during exploitation, which are of the same type as the specimens that where analysed experimentally and numerically. Shown in Figure 6 is the representation of the stress state in the cross-section along the *y*-axis of the prosthesis, i.e. its interior.

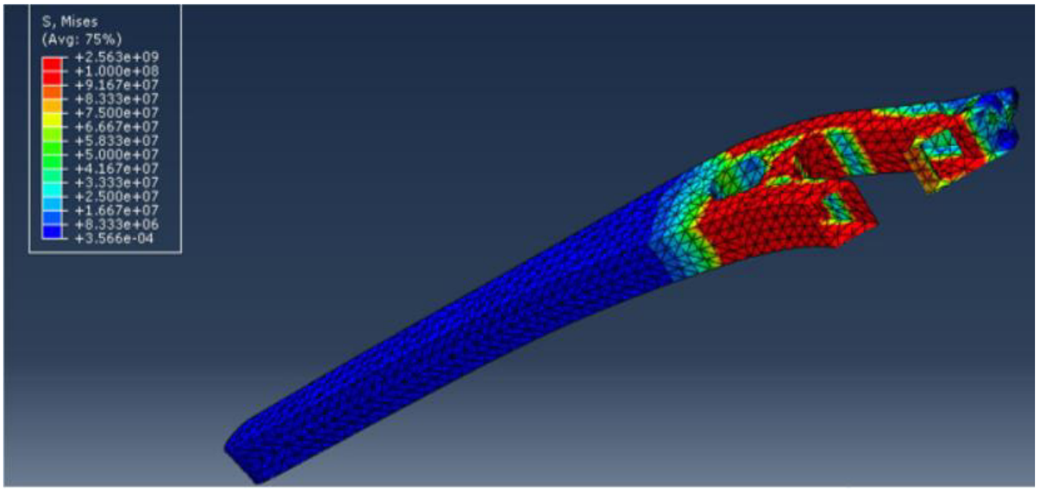

Figure 5. Stress state in the hole cross-section of the implant

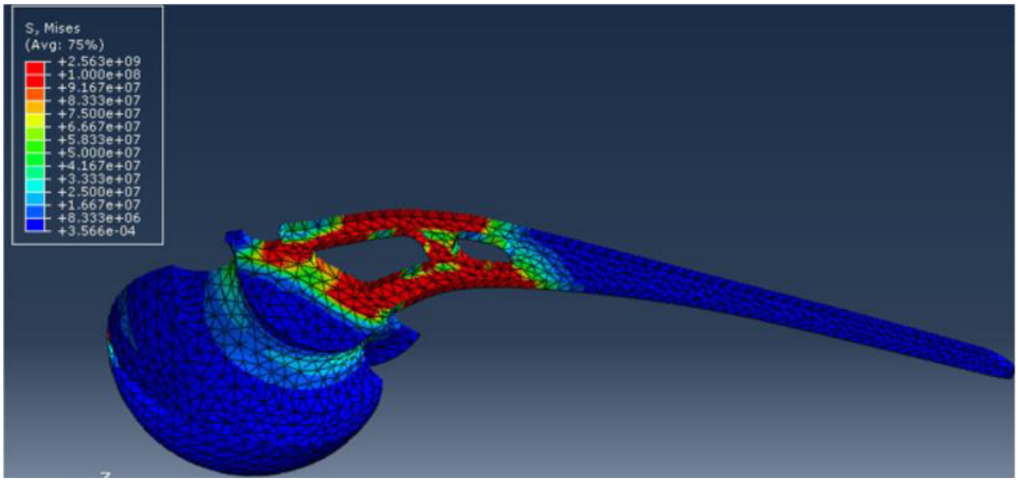

Figure 6. Stress state in the cross-section along the *y*-axis of the prosthesis

Shown in Figure 7 is the total prosthesis displacement under the effects of maximum load.

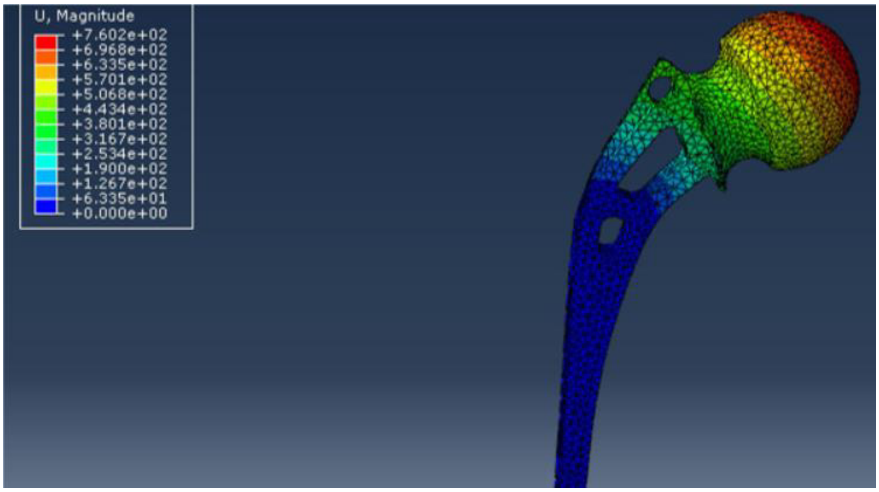

Figure 7. Total prosthesis displacement under the effects of maximum load

For the results obtained by static numerical analysis, values of Von Misses stresses are smaller than the yield strength of Ti-6Al-4V and Co-Cr alloy, but larger than the yield strength of 316L steel, even under highest assumed load magnitudes.

4. CONCLUSION

Experimental analysis was performed on three hip implant specimens removed from patients after revision surgery to determine mechanical behaviour under chosen critical stress load values, by using non-contact three-dimensional displacement and strain field measurement system. [3] During further study, numerical models were created, which gave results close to obtained experimental values, and a simulation of mechanical behaviour of hip prosthesis under characteristic loading was performed. Based on the results presented in this paper and obtained by applying the Finite Element Method, one can conclude the following:

- FEM is reliable and powerful tool for stress-strain analysis of complex-shaped implants, like the artificial hip
- Results of the numerical analysis show that the stress state in the implant cross-section with the hole corresponds to the location of the hole in the actual prosthesis, and it represents the location of critical stress concentration
- Stresses caused by maximum static loading are still lower than the yield strength of Ti-6Al-4V and CoCrMo alloy, but higher than the yield strength of 316L stainless steel.

The design objective for a hip stem is to have a low stress, displacement, and wear with a very high fatigue life. During the optimization process in search of ideal biomaterials and geometry for hip replacement implants preclinical tests must be carried out in order to check whether new models can guarantee the mechanical resistance to the physiological load. It can be concluded that the application of the finite element method (FEM) is a good alternative approach to provide a preliminary results and overview of the mechanical properties of potential implant models.

ACKNOWLEDGEMENT

We acknowledge the support for this investigation by Ministry for Education, Science and Technological Development, project TR 35040.

REFERENCES

[1] Park, J.B. “Hip Joint Prosthesis Fixation-Problems and Possible Solutions.” The Biomedical Engineering Handbook: Second Edition, CRC Press LLC, 2000, 46.1 - 46.3
[2] Jeffrey N Katz Total joint replacement in osteoarthritis. // Best Practice & Research Clinical Rheumatology, 20, 1(2006), pp. 145–153.
[3] Kaled Legweel, Aleksandar Sedmak, Katarina Čolić, Miloš Milošević, Nenad Mitrović, Žarko Mišković, Sergej Hloch, Experimental and numerical research of mechanical behaviour of titanium alloy hip implant, Technical Gazette ISSN 1330-3651 (print) ISSN1848-6339 (Online), Vol. 24/No. 3, 2017., DOI: 10.17559/TV-20160219132016
[4] Jesus Chao, Victor Lopez, „Failure analysis of a Ti6Al4V cementless HIP prosthesis“, Engineering Failure Analysis 14, pp. 822–830, 2007.

[5] S. Griza et al., „Design aspects involved in a cemented THA stem failure case“, Engineering Failure Analysis Volume 16, Issue 1, pp. 512–520, 2009.
[6] M. A. L. Hernandez-Rodriguez, J. A. Ortega-Saenz, G. R. Contreras-Hernandez, „Failure analysis of a total hip prosthesis implanted in active patient“, Journal of the Mechanical Behavior of Biomedical Materials 3, pp. 619–622, 2010.
[7] Manish Paliwal, D. Gordon Allan, Peter Filip, „Failure analysis of three uncemented titanium-alloy modular total hip stems“, Engineering Failure Analysis Volume 17, Issue 5, pp. 1230–1238, 2010.
[8] S. Griza, , M. Reis, Y. Reboh, A. Reguly, T.R. Strohaecker. Failure analysis of uncemented total hip stem due to microstructure and neck stress riser. Engineering Failure Analysis Volume 15, Issue 7, October 2008, Pages 981–988
[9] Sedmak A.; Čolić K.; Burzić Z.; Tadić S. Structural Integrity Assessment of Hip Implant Made of Cobalt-Chromium Multiphase Alloy. // Structural Integrity and Life, 10, 2(2010), pp. 161-164.
[10] David Bennett, Tarun Goswami, Finite element analysis of hip stem designs, Materials & Design, Volume 29, Issue 1, 2008, Pages 45–60
[11] A.Z. Senalp, O. Kayabasi, H. Kurtaran, Static, dynamic and fatigue behavior of newly designed stem shapes for hip prosthesis using finite element analysis, Materials and Design, 28, pp. 1577–1583, 2007.
[12] Piotr Niesłony, Wit Grzesik, Piotr Laskowski, Jan Sienawski. Numerical and Experimental Analysis of Residual Stresses Generated in the Machining of Ti6Al4V Titanium Alloy. 2nd CIRP Conference on Surface Integrity (CSI), Procedia CIRP, Volume 13, 2014, Pages 78–83
[13] E. Pyburn, T. Goswami, Finite element analysis of femoral components paper III–hip joints, Materials and Design, 25, pp. 705–713, 2004.
[14] N Nuño, M Amabili. Modelling debonded stem–cement interface for hip implants: effect of residual stresses. Clinical BiomechanicsVolume 17, Issue 1, January 2002, Pages 41–48
[15] Zohar Yosibasha, , , Alon Katza, Charles Milgromb, Toward verified and validated FE simulations of a femur with a cemented hip prosthesis. Medical Engineering & Physics. Volume 35, Issue 7, July 2013, Pages 978–987
[16] Hai-bo Jiang, Static and Dynamic Mechanics Analysis on Artificial Hip Joints with Different Interface Designs by the Finite Element Method Journal of Bionic Engineering Volume 4, Issue 2, June 2007, Pages 123–131
[17] Maroš Hrubina, Zdeněk Horák, Radek Bartoška, Leoš Navrátil, Jozef Rosina Computational modeling in the prediction of Dynamic Hip Screw failure in proximal femoral fractures Journal of Applied Biomedicine. Volume 11, Issue 3, 2013, Pages 143–151
[18] Makarand G. Joshi, Suresh G. Advani, Freeman Miller, Michael H. Santare, Analysis of a femoral hip prosthesis designed to reduce stress shielding, Journal of Biomechanics 33, pp. 1655-1662, 2000.
[19] S Gross, E.W Abel, A finite element analysis of hollow stemmed hip prostheses as a means of reducing stress shielding of the femur, Journal of Biomechanics, Volume 34, Issue 8, August 2001, Pages 995–1003
[20] Katarina Colic, Fracture behaviour analysis of artificial hip biomaterials, Doctoral thesis (in Serbian), University of Belgrade, Faculty of Mechanical Engineering, 2012
[21] PJ Prendergast Clinical Finite element models in tissue mechanics and orthopaedic implant design Biomechanics Volume 12, Issue 6, September 1997, Pages 343–366
[22] J. Geringer, L. Imbert, K. Kim Computational Modelling of Biomechanics and Biotribology in the Musculoskeletal System, 13 – Computational modeling of hip implants, Biomaterials and Tissues, Woodhead Publishing Limited, 2014, Pages 389–416