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

1877-7058 © 2016 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/4.0/).
Peer-review under responsibility of the organizing committee of ICMEM 2016
doi:10.1016/j.proeng.2016.06.664

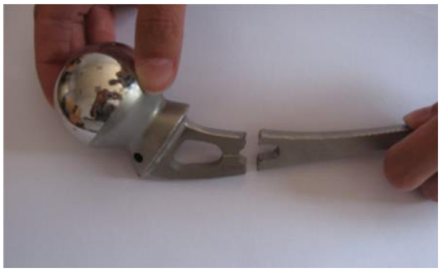

Figure 1. Artificial hip prosthesis failure

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.

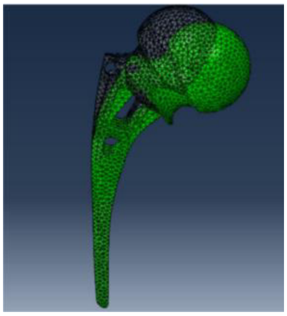

Figure 2. Comparative representation of the initial state and maximum load state of the implant