Finite Element Analysis of the Modified Hip Implant Surface

Aleksandra Vulović[1,2,3*], Fernando Warchomicka[4], Claudia Ramskogler[4], Christof Sommitsch[4], Nenad Filipović[1,2,3]

[1] *Faculty of Engineering, University of Kragujevac, Sestre Janjić 6, 34000 Kragujevac, Serbia*
[2] *Bioengineering Research and Development Center, Prvoslava Stojanovića 6, 34000 Kragujevac, Serbia*
[3] *Steinbeis Advanced Risk Technologies Institute doo Kragujevac, Kneza Milosa 25, Kragujevac, Serbia*
[4] *University of Technology, Institute of Materials Science, Joining and Forming, Kopernikusgasse 24, 8010 Graz, Austria*

Abstract:
Hip joint has an important role in the human body. It connects the lower and upper parts of the body and provides stability during daily activities such as walking, running, etc. After the hip has been damaged by an illness such as arthritis (usually osteoarthritis, rheumatoid arthritis or traumatic arthritis) or by a fracture, the hip replacement surgery is often performed. This is one of the most successful procedures. It helps with pain and disability reduction and with joint motion increase. During this procedure, the damaged bone and cartilage are removed and replaced with prosthetic components. Femoral head is removed and instead of it, metal stem is inserted into the femur. When this implant is inserted into the femur, new bone that begins to form interlocks with the hip implant. This process is commonly analyzed using *in vivo* experiments on animals, and not many papers deal with numerical approach. The present paper aims to apply the Finite Element Method (FEM) to analyze and determine the effect of a modified surface in a hip implant of titanium alloy Ti-6Al-4V. For the numerical simulations, three-dimensional model that consists of four layers (implant surface; bone tissue with highly reduced mechanical strength; bone tissue with slightly reduced mechanical strength; normal strength bone) was used. A 3D profile of a designed topography was added on the surface of the implant. The analyzed topography was produced using electron beam (EB) technique on the surface of titanium alloy, which showed a good response of osteoblast cells. Roughness was analyzed by infinite focus microscopy and the whole topography exported for FEM analysis. FEM simulations provide information about the behavior of topography under a wide range of loading conditions. This means that we could reduce the number of implants inserted into animals by choosing the appropriate topographies after FEM approach. With a growing number of hip replacement procedures, this will become an important issue as choosing the best topography could potentially improve life quality.

Keywords:
hip implant, modified surface, finite element method

Acknowledgement:
Research supported by the grants: European Virtual Institute on Knowledge-based Multifunctional Materials (KMM-VIN) Research Fellowship at the TU Graz, Austria, European Project H2020 PANBioRA 760921-2, III41007 and OI174028 grants from Ministry of Education, Science and Technological Development of Serbia.

*Corresponding author, e-mail: aleksandra.vulovic@kg.ac.rs