Simulation of the Interlocking Capacity of the Modified Hip Implant Surface

Aleksandra Vulović[*, **], Fernando Warchomicka[***], Claudia Ramskogler[***], Christof Sommitsch[***], Nenad Filipović[*, **]

[*] Faculty of Engineering, University of Kragujevac, Kragujevac, Serbia
[**] Bioengineering Research and Development Center, Kragujevac, Serbia
[***] Graz University of Technology, Graz, Austria

aleksandra.vulovic@kg.ac.rs
fica@kg.ac.rs
fernando.warchomicka@tugraz.at
claudia.ramskogler@tugraz.at
christof.sommitsch@tugraz.at

Abstract— **During hip replacement procedure femoral head is removed and instead of it implant is inserted into the hollow femur. When an implant is inserted into the fractured bone the healing process starts to happen, which means that the newly formed bone interlocks with the inserted implant. Interlocking capacity is commonly analyzed using in vivo experiments, and only several papers have information on numerical approach. The goal of this study was to use the numerical approach to analyze the interlocking capacity of the modified hip implant surface. Numerical approach included implementation of the finite element method (FEM) for simulation of the interlocking capacity of modified surface topographies in the titanium alloys (Ti-6Al-4V) implants. Analyzed topographies were produced by the electron beam (EB) technique on the surface of the titanium alloy. The structures were analyzed by infinite focus microscopy and exported as a 3D profile for the application of the finite element method. The biggest advantage of this approach is that it can provide information about different types of topographies under a wide range of loading conditions before needing to insert an implant into an animal.**

I. INTRODUCTION

The hip is a ball-and-socket joint between the upper end of the femur and the pelvis. It connects the lower and upper parts of the body, while bearing loads up to 5 times of bodyweight [1]. Also, the joint provides stability during daily activities (e.g. walking, running, etc.). The geometry of the hip allows rotation in all directions while muscles provide with forces that are needed for joint movement and help with stability [2].

Human life expectancy has been increasing in the last decades. With aging we are facing lot of health challenges. Some of those, such as arthritis causes disability and chronic hip pain. In those situations, hip replacement surgery is necessary. Hip replacement surgery is one of the most successful procedures that restores patient mobility. Also, it is a procedure with a minimum rate of early complication while providing immense improvement to the quality of life [3]. It is performed when bone and cartilage need to be removed as a result of a degenerative process or a trauma to a hip joint. In those situations, bone and cartilage are replaced with artificial joint.

The femoral head that is damaged is removed and is replaced with a metal stem that is inserted in the center of the femoral bone. On the upper part of the stem, a ball (ceramic or metal) is added so that it can replace removed damaged femoral head. Cartilage that is damaged is removed as well, and instead a metal socket is placed. In the end a spacer (metal, plastic or ceramic) is inserted between the socket and the ball in order to provide better movements.

When an implant is inserted into the fractured bone, the healing process starts to happen. This healing process is similar to primary bone healing. Osseointegration is a direct functional connection between bone and inserted implant. After the osseintegration is established, along the fixture surface cortical bone is formed [4]. One method to improve mechanical interlocking between bone and implant and enhance osseintegration is to use implant that has rough surface [5, 6]. The fixation of osseointegrated implants depends on the mechanical interlocking [7].

Interlocking capacity is commonly analyzed using in vivo experiments, and not many papers deal with numerical approach. Results from the in vivo studies indicate that increased surface roughness of cylindrical implants leads to increased interfacial shear strength [8]. Effect of surface texturing of orthopedic implants has been experimentally investigated in previous years. Ito et al. [9] have investigated possibility of polyethylene wear reduction by forming concave dimples on the surface of the metal femoral head while Zhou et al. [10] analyzed influence of concave dimples on the metallic counterface on the wear of ultra-high molecular weight polyethylene. Effect of surface texturing on the elastohydrodynamic lubrication analysis of metal-on-metal hip implants was analyzed by Gao et al. [11].

Mathematical model presented in [7], showed that rough surface led to increase in interfacial shear strength. One of the papers with numerical approach [5], showed a shear stress distribution for a non-resorbable fibre-reinforced composites during static load while bone and implant were rigidly bonded. Improved approach can be seen in [6], where better boundary conditions and geometry were used.

The goal of the present study was to obtain stress values for the modified hip implant surface using finite element method and use these values for assessment of interlocking capacity. This approach has been used in design and analysis of orthopedic devices, as this approach complements experimental work and can provide information that cannot be obtained from experiments.

The rest of the paper is organized as follows. The surface modification and explanation of the model used for finite element method are included in the part II. Part III covers obtained results while conclusion is given in part IV.

II. Materials & Methods

A. Electron Beam Technique

Modified surface topographies were produced by highly energetic electron beam (EB) technique on the surface of the titanium alloy (Ti6Al4V). Modification was performed using an electron beam welding (EBW) machine, model Probeam EBG 45-150 K14. Sample plates had dimensions 15 × 15 × 2 mm^3. This technique was successfully applied for titanium alloys structuring for biomedical application [12].

Created surface was analyzed using Alicona Infinite Focus microscope. The modified surface had to be exported as a 3D profile in order to use it for Finite Element Analysis. This was done with special focus variation technique. Acquired surface is shown in Fig. 1.

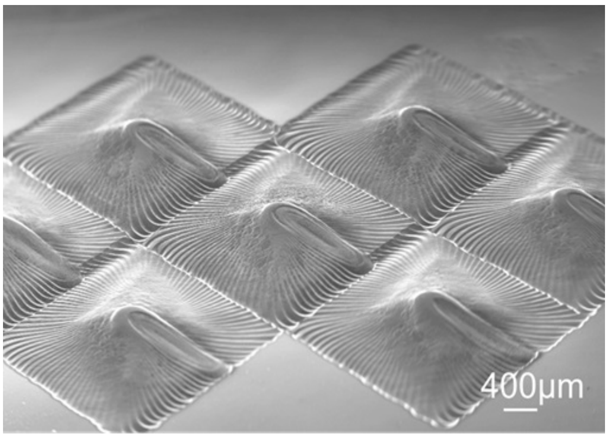

Figure 1. EB Structuring of Ti-6Al-4V with a square array with 100 arms each figure.

B. Finite Element Analysis

In order to carry out computational simulations of bone-implant interaction, the finite element models of implant and bone were modeled. The created three-dimensional model (Fig. 2) had four layers. The layers were created using a computer aided design (CAD) software. Each layer was exported from CAD software as .stp file in order to use it for mesh creation. Surface mesh was automatically generated and manually refined in order to improve mesh quality and reduce sharped triangular elements. Models created in CAD software had smooth surface.

Next step in model creation included manually adding previously created and exported modified surface on top of the implant (Fig. 3).

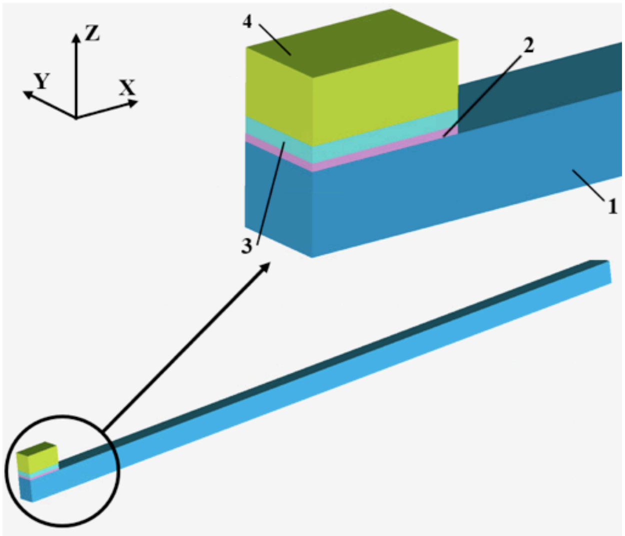

Figure 2. Created three-dimensional model

Modified surface was added to the end of the implant and its position will be of interest in future studies.

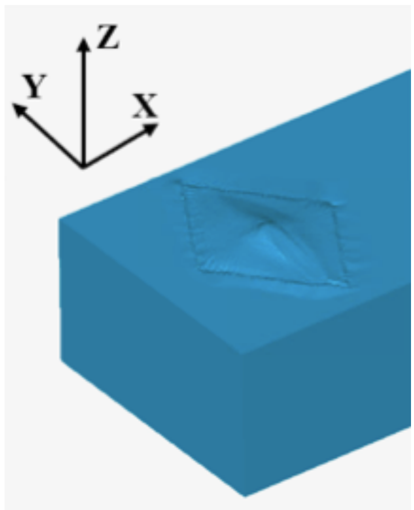

Figure 3. Implant (layer 1) after manually adding modified surface from Fig 1.

The implant surface (layer 1 – Fig. 2) was in contact with the very thin layer of the bone tissue with highly reduced mechanical strength (layer 2), followed by layer of the bone with the slightly reduced mechanical strength (layer 3). The last layer represented a healthy cortical femur bone (layer 4). Bone layers were modeled only to cover part of the implant where rough surface was added. The rest of the implant has smooth surface and it is of no interest at the moment.

This model consisted of 146453 nodes and 667527 tetrahedral elements.

Materials used for these calculations are considered to be homogenous, linear elastic and isotropic. Material properties (Young's modulus and Poisson's ratio) for all layers were gathered from the literature. Values are shown in Table 1. Titanium Alloy (Ti-6Al-4V) was chosen for implant material. Although material properties of bones, especially cancellous bone, are really complex, due to the computer constraints we had to consider all the bones to be homogenous, linear elastic and isotropic. This was done in order to obtain results in less time. In future studies we will use improved bone material properties.

TABLE I.
MATERIAL PROPERTIES – LAYERS 1, 2 AND 4.

	Young's modulus [GPa]	Poisson's ratio	Reference
Layer 1	109	0.34	[13]
Layer 2	0.02	0.3	[14]
Layer 3	10	0.3	[14]
Layer 4	16.7	0.3	[14]

Applied boundary conditions included load and several constraints. For load we have used maximum force value on the hip joint during the gait cycle. This value depends on a patient and for this analysis we have chosen the average weight of 70kg.

Applied constraints were adapted from paper [6], while taking into consideration the difference in model axis. The upper surface of layer 4 was fixed, while the sides of the implant were only allowed to move in x direction (locked in y and z direction) and lower surface of the implant was locked in the z direction. All other elements were allowed to move in all three directions.

Micro motions can occur between bone and implant. These motions can cause wear of the material. In order to simulate this accurately we have defined friction coefficient between layers 1 and 2. We have used friction coefficient of 0.39 obtained from the experimental study where tribological behavior of Ti6Al4V was investigated against cow bone [14]. The contact between other layers was considered to be glued.

After models were created and material properties and boundary conditions applied we were able to perform numerical simulations. Simulation was performed in order to analyze healing process.

III. RESULTS & DISCUSSION

Results depend on the quality of the created 3D model, elements, material properties and boundary conditions.

Fig 4. shows the von Mises stress distribution for the modified surface.

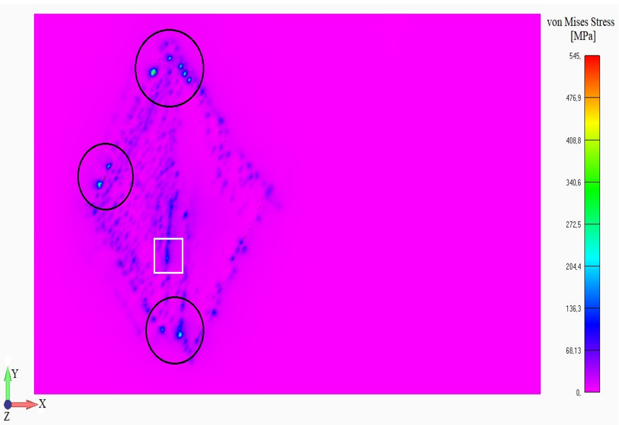

Figure 4. von Mises stress distribution for the modified surface.

It can be noticed that modified surface has no effect on the von Mises stress values of the surrounding structure. In the Fig. 4 there were several small areas (marked with black circles) with high stress values, up to 545 MPa. The reason for this was that modified surface was additionally added to the implant surface, which led to several elements with bad quality. As it can be seen these areas were on the edges of the surfaces and have not been changed as we wanted to keep profile the way it was exported.

If these areas are not taken into consideration, as they are most likely result of manual combining of implant model with modified surface mesh, the highest value was around 220 MPa. This value was calculated in the area represented by white square (Fig. 4).

Fig. 5 shows the contact pressure for the modified surface.

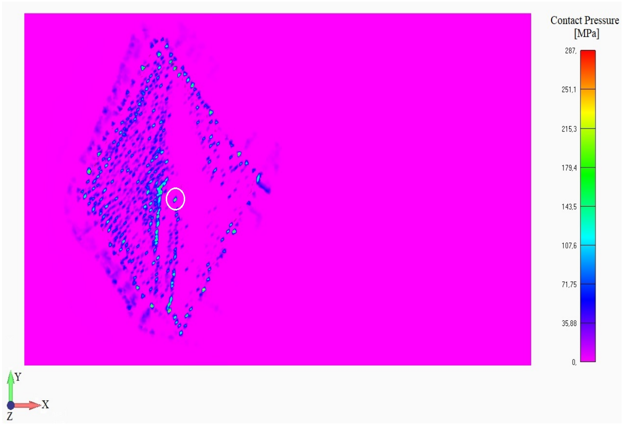

Figure 5. Contact pressure for the modified surface.

Again, it can be noticed that modified surface has no effect on the results of the surrounding structure. As with the previous case, the same areas have higher contact pressure values (up to 287 MPa) compared to the rest of the modified surface. The highest value for the contact pressure (white circle in Fig. 5) was around 205 MPa. It is important to notice that the left half of the surface had higher contact pressure values compared to the right half. This will be further analyzed.

A few papers have analyzed surface modification using FEM. It is hard to compare results as this modified surface is very complex. We were able to compare von Mises stress values outside of our modified surface. Fouad [16] had tried to predict stress of the bone fracture. Values he obtained for von Mises stress at fracture site at 1% healing (corresponds to our layer 2) was slightly higher compared to our values in layer 2. We had obtained about 2.3 MPa (outside of the modified surface), while he obtained 2.7 MPa. However, von Mises stress values in the area that was in the contact with modified surface showed significantly higher stress values, up to 130 MPa. These values require running more simulation using different boundary conditions in order to better understand it.

Obtained shear stress was significantly higher in the area of modified surface compared with smooth implant area. It is in range from 1 MPa (smooth area) to 100 MPa (modified surface). According to Mattila et al [5], shear stresses at the implant-bone interface should be minimized in order to promote bone ingrowth.

In order to improve interlocking capacity, we need to reduce shear stresses. There are several ways in which we can improve obtained results. First step will be to refine mesh near the edges of modified surface in order to reduce stress values. Friction coefficient value is also parameter that can be further analyzed as well as different boundary conditions combined with more accurate representation of material properties. Beside these parameters we also plan to analyze how multiple modified surfaces instead of one (presented in this paper) influence stress distribution.

IV. CONCLUSION

The presented results are just first step in our study. The obtained results show promise in this approach and have left some questions open. Further analysis is needed to determine how the location of the surface modification and its size affect stresses for different loads conditions as well as interlocking capacity.

In the industry of orthopedic implants, there is a growing need for simulations that will represent daily situations. FEA could be implemented to analyze stress in different implant topographies. This way we are able to analyze different types of surface modifications in order to determine which one has the best characteristics, before needing to insert implant into a rabbit or any other animal. Based on the information we could potentially choose the best topography for each type of implant (i.e. hip, dental, etc.) and improve the bone healing process.

ACKNOWLEDGMENT

The research has been carried out with support of the European Virtual Institute on Knowledge-based Multifunctional Materials (KMM- VIN) Research Fellowship at the Institute of Materials Science and Technology (TU Graz, Austria) 1.09-30.09.2016; Ministry of Education, Science and Technological Development, Republic of Serbia with projects III41007 and OI174028. C. Ramskogler acknowledges the government of Styria for the financially support, within the project "BioTiMod" in the framework of HTI_Tech for Med. The EB equipment is sponsored by the European Fond of Regional Development (EFRE), government of Styria and Graz University of Technology.

REFERENCES

[1] N. Hashimoto et al., "Dynamic Analysis of the Resultant Force Acting on the Hip Joint During Level Walking," *Artificial Organs*, vol. 29, pp. 387-392, 2005.

[2] D. Byrne, K. Mulhal, J. Baker J, "Anatomy & biomechanics of the hip," *The Open Sports Medicine Journal*, Vol. 4, pp. 51-57, 2010.

[3] H. Derar, M. Shahinpoor, "Recent patents and designs on hip replacement prostheses," *The Open Biomedical Engineering Journal*, Vol. 9, 92-102, 2015.

[4] R.S. Jayesh, V. Dhinakarsamy, "Osseointegration," *J Pharm Bioallied Sci.*, Vol. 7 (Suppl 1), p. S226.

[5] R. H. Mattila et al., "Bone attachment to glass-fibre- reinforced composite implant with porous surface," *Acta biomaterialia*, Vol. 5, pp.1639-1646, 2009.

[6] A. Halldin, M. Ander, M. Jacobsson, S. Hansson, "Simulation of the mechanical interlocking capacity of a rough bone implant surface during healing," *BioMed Eng OnLine*, Vol. 14, 45, 2015.

[7] S. Hansson, M. Norton, "The relation between surface roughness and interfacial shear strength for bone-anchored implants. A mathematical model," *Journal of Biomechanics*, Vol. 32, pp. 829-836, 1999.

[8] J.M.I. Loberg, S. Hansson, E. Ahlberg, "Characterisation of titanium dental implants I: critical assessment of surface roughness parameters," *TOBIOMTJ*. Vol. 2, pp. 1–8, 2010.

[9] H. Ito, K. Kaneda, T. Yuhta, I. Nishimura, K. Yasuda, T. Matsuno, "Reduction of polyethylene wear by concave dimples on the frictional surface in artificial hip joints," *The Journal of Arthroplasty*, Vol. 15, pp. 332-338, 2000.

[10] X. Zhou, A.L. Galvin, Z. Jin, X. Yan, J. Fisher, "The influence of concave dimples on the metallic counterface on the wear of ultra-high molecular weight polyethylene," *Proc. Inst. Mech. Eng., Part J: J. Eng. Tribol*. Vol. 226, pp. 455–462, 2012.

[11] L. Gao, P. Yang, I. Dymond, J. Fisher, Z. Jin, "Effect of surface texturing on the elastohydrodynamic lubrication analysis of metal-on-metal hip implants," *Tribology International*, Vol. 43, pp. 1851-1860, 2010.

[12] C. Ramskogler, F. Warchomicka, S. Mostofi, A. Weinberg, C. Sommitsch, "Innovative surface modification of Ti6Al4V alloy by electron beam technique for biomedical application," *Mater. Sci. Eng. C*, Vol. 78, pp. 105–113, 2017.

[13] K.S.S. Aradhya, M.R. Doddamani, "Characterization of Mechanical Properties of SiC/Ti-6Al-4V Metal Matrix Composite (MMC) Using Finite Element Method," *American Journal of Materials Science,* Vol. 5(3C), pp. 7-11, 2015

[14] S. Das, S.K. Sarangi, "Finite Element Analysis of Femur Fracture Fixation Plates,", *International Journal of Basic and Applied Biology*, Vol. 1(1), pp. 1-5, 2014.

[15] S. Mischler, G. Pax, "Tribological behavior of titanium sliding against bone," *Eur Cell Mater Suppl.* Vol. 1(3), pp. 28–9, 2002.

[16] H. Fouad, "Assessment of function-graded materials as fracture fixation bone-plates under combined loading conditions using finite element modelling," Medical Engineering & Physics, Vol. 33(4), pp. 456-463, 2011.