Application of Numerical Methods in Design and Analysis of Orthopedic Implant Integrity

Katarina Čolić[1(✉)], Aleksandar Grbović[2], Aleksandar Sedmak[2], and Kaled Legweel[3]

[1] Innovation Center of Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia
`kbojic@mas.bg.ac.rs`
[2] Faculty of Mechanical Engineering, University of Belgrade, Belgrade, Serbia
[3] Sebha High Institute, Sebha University, Sabha, Libya

Abstract. In this paper a numerical analysis of hip implant model and hip implant model with a crack in a biomaterial is presented. Hip implants still exhibit problem of premature failure, promoting their integrity and life at the top of the list of problems to be solved in near future. Any damage due to wear or corrosion is ideal location for crack initiation and further fatigue growth. Therefore, this paper is focused on integrity of hip implants with an aim to improve their performance and reliability. Numerical models are based on the finite element method (FEM), including the extended FEM (X-FEM). FEM became a powerful and reliable numerical tool for analysis of structures subjected to different types of load in cases where solving of these problems was too complex for exclusively analytical methods. FEM is a method based on discretization of complex geometrical domains into much smaller and simpler ones, wherein field variables can be interpolated using shape functions. Numerical analysis was performed on three- dimensional models, to investigate mechanical behaviour of a hip implant at acting forces from 3.5 to 6.0 kN. Short theoretical background on the stress intensity factors computation is presented. Results presented in this paper indicate that acting forces can lead to implant failure due to stress field changes. For the simulation of crack propagation extended finite element method (XFEM) was used as one of the most advanced modelling techniques for this type of problem.

Keywords: Hip replacement implant · Stress intensity factor · Crack growth
Numerical simulations · Extended finite element method

1 Introduction

In the field of orthopaedic surgery, it is very important to determine the stress-strain condition of the implants and bones. Mechanical loads of organs and tissues within a human body cause stresses and strains. Strain caused by mechanical load produce internal mechanical forces within the body. For simpler geometric shapes, analysis of mechanical behaviour can be determined analytically, wherein for complex cases, it is necessary to apply a numerical method, such as finite element method (FEM). Finite element analysis, represents a numerical analysis used for solving of complex geometry

problems, for which obtaining of an analytical solution is extremely difficult. FEM is a method based on discretization of complex geometrical domains into much smaller and simpler ones, wherein field variables can be interpolated using shape functions.

Application of FEM in biomedical applications is becoming more and more frequent, causing complications in terms of complicated geometry, when it is almost impossible to reach an analytical solution. During previous studies, this method was used to determine the stress state of the hip implants, as well as on orthopedic plates of different geometries [1–3].

In FEM, a complex region which defines the continuum is discretized into simple geometric shapes - elements. It is assumed that these elements have properties and relations which can be mathematically expressed as unknown quantities in certain points of elements - nodes. A process of connecting and combining of individual elements in a given system is applied. After taking into account the influence of load and boundary conditions, a series of linear or non-linear equations is typically obtained. Solving of these equations provides an approximate behaviour of a continuum or a system. The algorithm for this method consists of the following steps: continuum discretization, selection of interpolation functions, calculating of system properties, forming of algebraic equations, solving of algebraic equation systems, and calculation of necessary influences in nodes for individual finite elements.

The advantages of applying FEM include: it is applicable to complex geometries, complex types of analysis, complex loads, and models made of non-homogeneous materials, etc. Types of errors that can occur include discretization errors, as well as formulation and numerical errors. The basic equation of FEM for static load conditions is $\{F\} = [K] \times \{u\}$, where $[K]$ is the general or global stiffness matrix, $\{u\}$ is the global displacement vector [4, 5].

FEM saw its first use in orthopaedic biomechanics in 1972, for the purpose of assessing of stresses in human bones. Since then, this method has been applied with increasing frequency in stress state analysis of bones and prosthetics, as well as fracture fixation, [1–3, 6–15], an example model is presented in Fig. 1 [1].
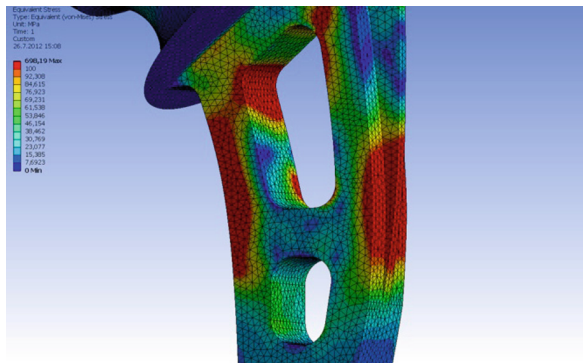
Fig. 1. Stress distribution on a hip implant obtained by the FEM, [1]

In addition to bones, this method can be used in analysis of numerous other tissues and organs. In case of orthopaedics, there has always been a significant interest in stress and load. However, mathematical tools available for stress analysis in classic mechanics were not suitable for calculations of extremely irregular structural properties of bone. Hence, the use of FEM represented a logical step due to its unique ability to determine stress state in structures with complex shape, load and material behaviour, [14–19]. In this paper, the basics of finite element method and its application to prosthetics will be presented, with particular attention to modelling of a hip replacement prosthesis.

2   Application of FEM in Analysis of Behavior of Biomaterials with Cracks

Modelling of fracture mechanics problems requires an adequate treatment of displacement and stress field singularity around the crack tip, where the biggest problem is reflected in drastic increase in discretization error, which occurs when using classic FE, such as the eight-node element. The most efficient solution is obtained by using the reduction technique (reducing the error to only 1%) or by applying special FE around the crack tip, which contains the strain field singularity [4, 5, 20–23].
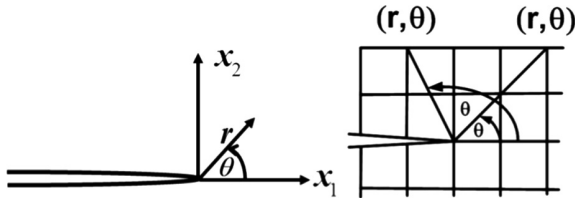

Fig. 2. Special crack tip elements.

In the case of displacement extrapolation method, Fig. 2, when crack propagation simulation in the material is performed using polar coordinates, the following expressions for equivalent coefficient can be defined for plane strain conditions, and it is determined according to expression (1):

$$K_I^* = \frac{\sqrt{2\pi}}{2} \frac{E}{(1+v)} \frac{1}{\sin\frac{\theta}{2}\left[2(1-v) - \cos^2\frac{\theta}{2}\right]} \frac{v(r,\theta)}{\sqrt{r}} \tag{1}$$

If the angle $\theta = 0$, the following expression is obtained:

$$K_I^* = \frac{\sqrt{2\pi}}{4} \frac{E}{1-v^2} \frac{v(r,\pi)}{\sqrt{r}} \tag{2}$$

$$K_I = \lim_{r\to\infty} K_I^* \tag{3}$$

Fracture mechanics parameters can be determined in a number of different methods, such us displacement extrapolation, J-Integral, stiffness derivative method, etc.

3 Application of X-FEM Method to Crack Growth Simulation

In order to evaluate the influence of initial defects in material on strength and life of structures, finite element analysis is applied to cracks of various shapes, sizes and locations. In these analyses, FEM is limited, since changes in crack topology require additional generating of mesh domain. This represents a significant constraint and complicates crack growth simulation on complex geometries. Extended finite element method (X-FEM) was developed in order to make calculations easier, which was required during positioning of arbitrary cracks within a finite element model [20, 24]. It is a very wide field of application of this method, first of all in determining the behavior of material with crack in the engineering, as well as in biomedical applications [25–27].

XFEM uses enhancement functions as a means of displaying all forms of discontinuous behaviour, such as crack displacement. Enhancement functions are introduced into the displacement approximation for only a small number of finite elements, relative to the size of the whole domain. Additional degrees of freedom are introduced for all elements where the discontinuity is present, and in some cases - depending on the type of the selected function - into adjacent elements, which are then referred to as mixed elements [28–31].

Displacement approximation can be expressed in the following way, by applying enhancement functions:

$$u^h(x) = \sum_i N_i(x)\left[u_i + \sum_j v^j(x)a_i^j\right] \tag{4}$$

The unity property is based on the fact that the sum of interpolation functions of finite elements equals one. Assuming that the unity property is fulfilled, additional enriching functions, i.e. improvement functions, can be given in displacement approximation. In this case, application of standard X-FEM displacement formulation approximates displacements as:

$$u(\xi, \eta, \zeta) = \sum_i N_i(\xi, \eta, \zeta)U_i + \sum_i N_i(\xi, \eta, \zeta)H(\xi, \eta, \zeta)b_i$$
$$+ \sum_i N_i(\xi, \eta, \zeta)\left(\sum_j \Psi_j(r, \theta)c_{ji}\right) \tag{5}$$

where $N_i(\xi, \eta, \zeta)$ are shape functions, $U_i \in \mathbf{R}^3$ are node displacement parameters for all nodes of a hexahedron element: $1 \sim 8$, $b_i \in \mathbf{R}^3$ are parameters of jump function on jump nodes, and $c_{ji} \in \mathbf{R}^3 \times \mathbf{R}^4$ are parameters of the branching function for nodes at the crack tip [28–31].

It is necessary during calculation to determine which mesh elements were divided by the crack and in which element the crack tip is located, taking into account that

X-FEM does not approximate the entire domain. In this sense, an unequivocal identification of elements uses two functions on the level of sets (LS functions), which are based on level set (LS) method.

Jump function H is defined as the sign of the level set φ:

$$H(\xi,\eta,\zeta) = \begin{cases} +1 : \varphi(\xi,\eta,\zeta) > 0 \\ -1 : \varphi(\xi,\eta,\zeta) < 0 \\ \pm 1 : \varphi(\xi,\eta,\zeta) = 0 \end{cases} \quad (6)$$

It should be noticed that function $H(\xi,\eta,\zeta)$ is not well defined when $\varphi(\xi,\eta,\zeta) = 0$, $H(\xi,\eta,\zeta) = \pm 1$ and $[\![H(\xi,\eta,\zeta)]\!] = 2$ merely represents a suitable way of calculating of the jump function in points which are located at the crack surface [28–31].

4 Numerical Calculation of Selected Hip Implant Models

Numerical models were made of a hip prosthesis in order to analyse material behaviour of an implant during load in an ideal case, and also when there is a crack in the material. In this sense, simplifications of problems related to implanting of the prosthetic were performed in order to fulfil the requirements in terms of size. In a realistic case, there are many factors that influence the integrity of the prosthetic, such as state of bones, effects of corrosion and biocompatibility of selected metallic biomaterials [32–35], as well as bone cement properties [36, 37], but it is not possible to simulate all of these effects, since they depend on individual cases as well [38].

For FEM analyses, performed in commercial software ABAQUS, three-dimensional models of the prosthetic and stem were made based on real prosthetic components, one example presented in a Fig. 3.

Fig. 3. Hip implant model made of CoCrMo alloy.

Selected model is made of CoCrMo alloy [33, 39], whose properties are given in Table 1.

Table 1. Mechanical properties

Properties	CoCrMo
Tensile strength	655
Yield point (0.2%), MPa	450
Elongation, %	8
Reduction area, %	8
Fracture point, MPa√m	310

4.1 Development of Numerical Models

The geometry of the stem has a significant effect on prosthesis performance. Stem with a smooth surface, generally speaking, reduces the stress concentration and enable significant fatigue resistance. Stem with a sharp or rugged surface enables good connection and prevents potential sliding of the joint. Level of stress concentration and tendency to fatigue fracture depend on the roughness of the stem surface.

A dimension of adopted geometrical model is given in the Fig. 4.
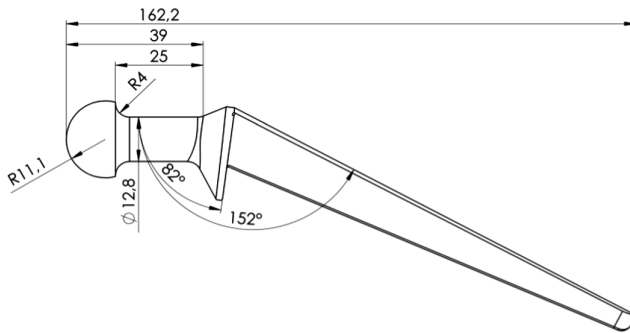
Fig. 4. Hip implant model – dimensions

The constitutive relation which is chosen for the given problem requires the specification of two constants, Young's elasticity modulus and Poisson's ratio.

Values of coefficients used for this problem in an example presented here, are given in Table 2.

Table 2. Values of coefficients used

Material	Elasticity modulus (GPa)	Poisson's ratio
CoCrMo	234	0.3

FEM analysis requires a numerical description of all external loads affecting the structure (points in which they act, magnitude, direction). These loads are usually

variable and are not always precisely defined, thus when using FEM analysis, a frequent question is which approach to use in order to obtain useful information.

Certain approximations have been introduced regarding boundary conditions and loads of the model. Taking into account that only the behaviour of metal structures is analysed in terms of crack presence in the biomaterial, and based on the review of relevant literature in terms of the expected location of crack initiation, it was possible to introduce a fixed support approximation for the load bearing structure of the implant in the lower part which is in contact with the bone.

Real load which acts on the implant was introduced in two ways, as compressive load on prosthetic cup, and as a force which acts in a single point on the acetabular part of the prosthetic. Loads defined in this way actually represent an approximation of real hip implant behaviour. Since real load acting on the hip is highest in the simulated direction, and in the first approximation of the bone-implant connection, a completely rigid connection can be assumed.

Two characteristic load types during the walking stage were selected, shown in Table 3, for normal and fast walking. In that sense, the values of the load range from minimum 4.9 BW to maximum 7.6 BW loads. For the purpose of numerical calculation, a person with a weight of 80 kg was selected, thus the expected loads on the implant during normal walking were 3845.5 N and for fast walking 5964.5 N respectively.

Table 3. The maximum force on the hip joint

Activity	The maximum force on the hip joint (a multiple of body weight)
Walking on flat surface	
Slow	4.9
Normal	4.9
Fast	7.6
Climbing the stairs	7.2
Coming down the stairs	7.1
Climbing the steep surface	5.9
Coming down the steep surface	5.1

## 4.2   Discretization of Structure in Numerical Models

For the purpose of FEM analysis, two types of finite elements were selected, which adequately describe the behaviour of complex three-dimensional structures, such as hip implants. Within the calculation, standard 3D stress elements type libraries were used, which are a part of the applied software package. Applied to all models were two types of 3D elements, an 8-node linear hex type element, with reduced integration and a 10-node tetrahedral type of quadratic element.

Discretization parameters for model calculation are shown in Table 4, and include the selected shape and number of elements and nodes.

Table 4. Discretization parameters of a numerical model

Model
Implant base:
Hexahedral type:(C3D8R)   8-node linear   Reduced integration
Implant cup:
Hexahedral type:(C3D8R)   8-node linear   Reduced integration   *Tetrahedral type*: (C3D10)   10-node quadratic
Total number of elements: 7 253
Number of nodes: 9 708

The finite elements mesh generated on the calculation modelis shown in Fig. 5.
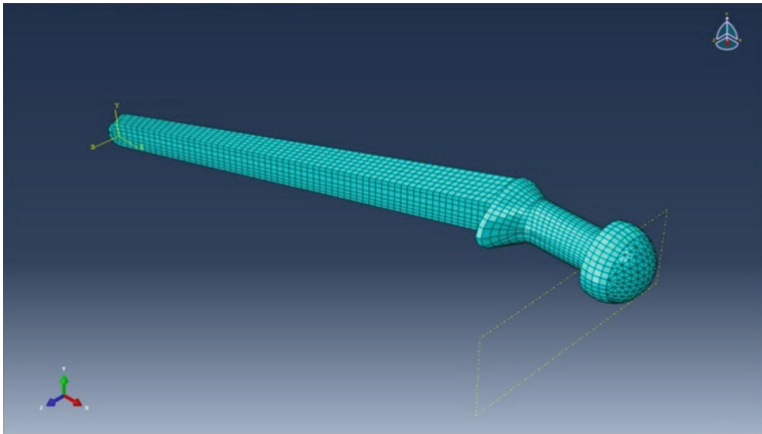
Fig. 5. Numerical model - FE mesh

5 Development of a Model with a Crack

The most recent method which is applied to modelling of cracked material behaviour is based on extended finite element method, X-FEM. [24–27].

Based on literature analysis, it was assumed that in places where stem was connected to the acetabular part, wear or corrosion can occur in material, and as a result of that, material damage may occur, i.e. the appearance of cracks [33, 40–42].

During further model development and preparation for FEM analysis, generating of the mesh on the implant model was performed, with an initial crack and a tendency to obtain a mesh as fine as possible.

Shown in Fig. 6 is the implant model with a generated finite elements mesh and an initial crack positioned in the critical area.
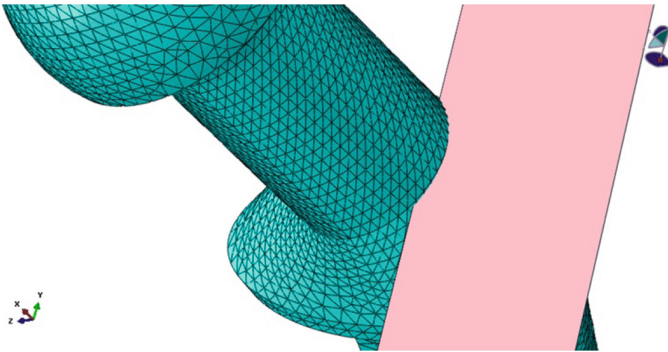

Fig. 6. Display of a discretized implant model with a crack in the critical area

6 Results of Numerical Analysis

To investigate the difference between results for loads defined by standard defined and real loads on implants that can appear in practice, it is necessary to analyze the prosthesis under a body weight static loading, and under the maximum load that can occur during the walking cycle.

For numerical model the distribution of von Mises stressis shown in Fig. 7, while applied loads is 3845.5 N on the selected implant surface that is 422 mm^2. For this model a standard element library is used, and the model is made using the C3D8R and
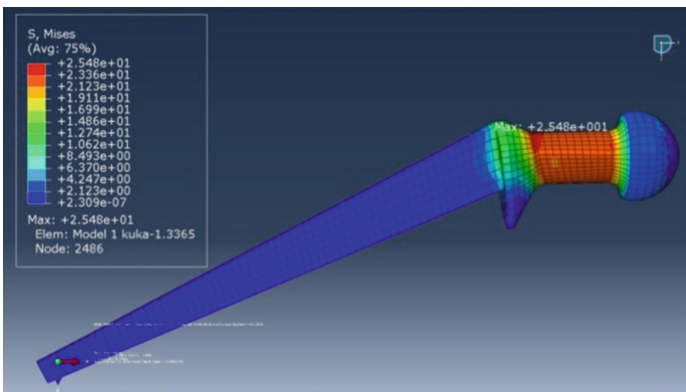

Fig. 7. Von Mises stress distributionon a numerical model.

C3D10 element types. Total number of elements is 7.253 and total number of nodes is 9.708. For this analysis total CPU time is 7.300 and deformation scale factor for graphical representation is 500.

Based on the stress field analysis for the characteristic loads, maximum stress values on an implant model are obtained, with two different types of load structure presented in this paper. The results are shown in Table 5.

Table 5. Results of numerical analysis

Activity	Load on implant (N)	Von Mises stress value (MPa)
Normal walking	3845.5	254
Fast walking	5964.5	387

Referring to the analysis of stress and strain state on the numerical model, it can be concluded that the area of maximum stress and strain values is right at the expected places on implant geometry. Numerical simulations on model show that stress concentration occurs in precisely those areas where in practice fatigue fracture occurred in implants of similar or same geometry.

With regard to these observations, further numerical analysis included numerical models of an implant with fatigue crack in the biomaterial, set just in spots of the highest stress concentration.

Crack propagation rate da/dN was obtained using Paris crack growth low, Paris exponent n is 2.28, and Paris coefficient is $2.05 \ 10^{-11}$.

Applied parameters for the numerical calculation of the fatigue crack propagation in biomaterial of the implant are given in Table 6.

Table 6. Numerical parameters for crack propagation behavior

Implant biomaterial
CoCr superalloy
Crack growth low:

$$\frac{da}{dN}=C(\Delta K)^n$$

Numerical calculation parameters:
n = 2.28
C = 2.056 10^{-11}
$Y_{ie.}$ = 1751
UTS = 1889
K_{Ie} = 3649
K_{IC} = 2780

The numerical analysis were done in numerical package Morpheo, which is based on application of extended finite element method, and is supported by numerical software for simulation and finite element analysis Abaqus.

The initial crack was set on a spot where occurrence of fatigue and the micro pitting it is expected.

Figure 8 shows the critical area on mono-block hip implant models in terms of fatigue crack appearance, i.e. the numerical model with crack inserted and finite element mesh generated, and as well the crack propagation in the material.
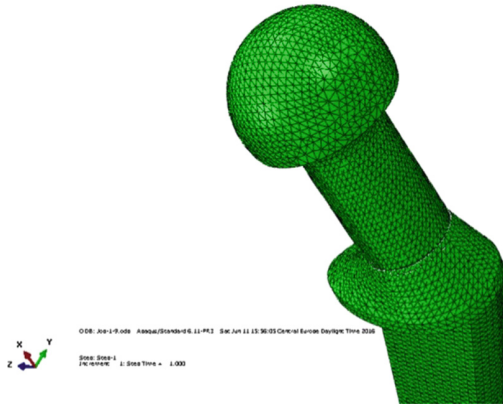

Fig. 8. Crack propagation on a numerical model.

Figure 9 shows the expansion of cracks in the material and stress distribution at the critical crack length for selected implant model.
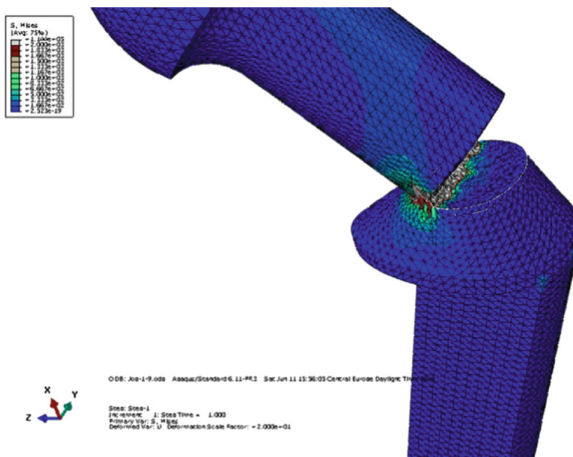

Fig. 9. Von Misses stress distribution on a crack surface

By applying numerical analysis on the model of an implant with crack fracture mechanics parameters were determined, i.e. value of the stress intensity factor K_I, K_{II} K_{III} i K_{ef}. It should be noted that all these values are determined for each calculation step. Based on theoretical considerations of the crack opening modes, it is clear that in this case of loading conditions on an implant, stress intensity factor values are much higher for the mode I than for modes II and III.

For each step of numerical calculation obtained stress intensity factors values K_I K_{II} and K_{III} are presented in Table 7.

Table 7. K_I values for numerical model

Calculation step	Crack length	K MPa√m		
		K_I	K_{II}	K_{III}
1	0.5	747.43	38.29	16.85
2	1	887.10	16.17	15.80
3	1.5	1042.62	21.98	7.93
4	2	1288.52	17.01	0.83
5	2.5	1556.34	16.65	6.69
6	3	1983.27	33.55	14.19
7	3.5	2716.15	37.63	22.28
8	4	3698.11	51.62	6.78
9	4.5	5806.34	109.64	33.78
10	5	10,447.68	93.51	1.16
11	5.5	27,257.73	989.75	193.49
12	6	114,211.8	191.97	1148.49

It is recommended to assume an initial crack in the material in accordance with the fracture criteria K_{IC}, when crack growth occurs in case of $K_I \geq K_{IC}$, i.e. the linear elastic analysis fracture criteria.

Obtained values suggest that the hip prosthetic with an initial crack, subjected to a normal walking, i.e. which works under walking cycle normal conditions, will have a number of cycles equal to 29,493 for numerical model before final fracture and prosthesis failure. Results were numerically obtained and show the number of cycles after the crack was initiated, and with the application of numerical simulation, the number of cycles for which the prosthetic functions under normal conditions was determined, assuming there are flaws in the material.

X-FEM had shown that it is an extremely efficient tool for numerical modeling of cracks in LEFM. Compared to standard FEM, X-FEM introduces significant improvements into numerical modeling of crack growth. Main advantages are that the finite element mesh does not need to be adjusted to crack boundaries (crack surface) in order to include geometric discontinuity, and that there is no need to regenerate the mesh in crack growth simulations.

It was shown that by applying modern numerical methods of biomaterial behaviour analysis it is possible to monitor three-dimensional crack behaviour in the material, as well as determining of characteristic fracture mechanics parameters.

7 Conclusions

Generally speaking, as shown and discussed in this paper, there are many influencing factors regarding integrity and life of hip implants. It is necessary to know precisely the loads, including variable stresses due to dynamic loading, biomaterial properties and corrosion behaviour, including fracture mechanics properties, and stress distribution focused on concentration areas. Therefore, new, modern methods are needed for thorough analysis of this significant problem, starting from advanced experimental methods, using fracture mechanics approach to estimate structural integrity, and advance FEM numerical simulations, including fatigue crack growth, both in experimental and numerical research.

(1) FEM is reliable and powerful tool for stress-strain analysis of complex shaped implants, such as the artificial hip. It has been shown that by applying modern numerical methods to biomaterial behaviour analysis, it is possible to monitor three-dimensional crack behaviour in a material, as well as to determine fracture mechanics parameters.

(2) Numerical simulation on developed models show that stress concentration occurs precisely in those areas, where in the examples from the practice there wasa fatigue fracture on the hip implant of very similar geometry. The areas on numerical models with maximum stress concentration coincide with the most common locations for the formation of cracks in biomaterials, which eventually lead to the weakening of the integrity of the prosthesis, or to failure.

(3) A finite element analysis was performed using three-dimensional models to examine the mechanical behaviour of hip prostheses at forces ranging from 3.5 to 6.0 kN. Results show that the force magnitudes acting on the implant are of interest, that according to implant biomaterial and design they can cause implant stress field changes, which can lead to structure integrity problems and implant failure.

(4) Extended finite element method (X-FEM) provided good agreement of experimental and numerical results for the fatigue crack growth.

Orthopaedic biomechanics problems are not examined easily in vivo due to inaccessible locations and invasive methods. In order to obtain clinically useful models, the next step should involve the development of numerical models of the human bones in which the implant is located. It should be noted that the construction of these models is not an easy task, i.e. to get the most accurate results the first step would be scanning the human hip in various patients to form a basis from which the numerical models would be formed. Future work might also include design of numerical models that will take into account the impact of compound between human bones and hip implants, as well as simulation of crack propagation in the cement biomaterial.

References

1. Colic, K., Sedmak, A., Legweel, K., Milošević, M., Mitrovic, N., Miškovic, Ž., Hloch, S.: Experimental and numerical research of mechanical behaviour of titanium alloy hip implant. Tehnicki vjesnik – Tech. Gaz. **24**(3), 709–713 (2017)
2. Tatić, U., Čolić, K., Sedmak, A., Mišković, Ž., Petrović, A.: Evaluation of the locking compression plates stress-strain fields. Tehničkivjesnik – Technical Gazette **25**(1), 112–117 (2018)
3. Milovanović, A., Sedmak, A., Čolić, K., Tatić, U., Đorđević, B.: Numerical analysis of stress distribution in total hip replacement implant. Struct. Integr. Life **17**(2), 139–144 (2017)
4. Reddy, J.N.: An Introduction to the Finite Element Method. McGraw-Hill, New York (2005)
5. Hutton, D.V.: Fundamentals of Finite Element Analysis. Mc Graw Hill, New York (2004)
6. Senalp, A.Z., Kayabasi, O., Kurtaran, H.: Static, dynamic and fatigue behavior of newly designed stem shapes for hip prosthesis using finite element analysis. Mater. Des. **28**, 1577–1583 (2007)
7. Colic, K., Sedmak, A., Grbovic, A., Tatic, U., Sedmak, S., Djordjevic, B.: Finite element modeling of hip implant static loading. Proc. Eng. **149**, 257–262 (2016)
8. Bennett, D., Goswami, T.: Finite element analysis of hip stem designs. Mater. Des. **29**(1), 45–60 (2008)
9. Pyburn, E., Goswami, T.: Finite element analysis of femoral components paper III-hip joints. Mater. Des. **25**, 705–713 (2004)
10. Paliwal, M., Gordon Allan, D., Filip, P.: Failure analysis of three uncemented titanium-alloy modular total hip stems. Eng. Fail. Anal. **17**(5), 1230–1238 (2010)
11. Gross, S., Abel, E.W.: A finite element analysis of hollow stemmed hip prostheses as a means of reducing stress shielding of the femur. J. Biomech. **34**(8), 995–1003 (2001)
12. Hrubina, M., Horák, Z., Bartoška, R., Navrátil, L., Rosina, J.: Computational modeling in the prediction of Dynamic Hip Screw failure in proximal femoral fractures. J. Appl. Biomed. **11**(3), 143–151 (2013)
13. Balac, I., Colic, K., Milovancevic, M., Uskokovic, P., Zrilic, M.: Modeling of the matrix porosity influence on the elastic properties of particulate biocomposites. FME Trans. **40**(2), 81–86 (2012)
14. Geringer, J., Imbert, L., Kim, K.: Computational Modelling of Biomechanics and Biotribology in the Musculoskeletal System, Computational Modeling of Hip Implants, Biomaterials and Tissues, pp. 389–416. Woodhead Publishing Limited, Sawston (2014)
15. Huiskes, R., Chao, E.Y.S.: A survey of finite element analysis in orthopedic biomechanics. First Decade J. Biomech. **16**(6), 385–409 (1983)
16. Prendergast, P.J.: Clinical finite element models in tissue mechanics and orthopedic implant design. Biomechanics **12**(6), 343–366 (1997)
17. Huiskes, R., Hollister, S.J.: From structure to process, from organ to cell: recent developments of FE-analysis in orthopaedic biomechanics. J. Biomech. Eng. **115**, 520–527 (1993)
18. Yosibasha, Z., Katza, A., Milgromb, C.: Toward verified and validated FE simulations of a femur with a cemented hip prosthesis. Med. Eng. Phys. **35**(7), 978–987 (2013)
19. Marangalou, J.H., Ito, K., van Rietbergen, B.: A new approach to determine the accuracy of morphology–elasticity relationships in continuum FE analyses of human proximal femur. J. Biomech. **45**(16), 2884–2892 (2012)
20. Belytschko, T., Black, T.: Elastic crack growth in finite elements with minimal remeshing. Int. J. Numer. Methods Eng. **45**(5), 601–620 (1998)

21. Moës, N., Dolbow, J., Belytschko, T.: A finite element method for crack growth without remeshing. Int. J. Numer. Methods Eng. **46**(1), 131–150 (1999)
22. Karihaloo, B.L., Xiao, Q.Z.: Modelling of stationary and growing cracks in FE framework without remeshing: a state-of-the-art review. Comput. Struct. **81**(3), 119–129 (2003)
23. Maligno, A.R., Rajaratnam, S., Leen, S.B., Williams, E.J.: A three-dimensional (3D) numerical study of fatigue crack growth using remeshing techniques. Eng. Fract. Mech. **77**(1), 94–111 (2010)
24. Kastratovic, G., Grbovic, A., Vidanovic, N.: Approximate method for stress intensity factors determination in case of multiple site damage. Appl. Math. Model. **39**(19), 6050–6059 (2015)
25. Kredegh, A., Sedmak, A., Grbovic, A., Sedmak, S.: Stringer effect on fatigue crack propagation in A2024-T351 aluminum alloy welded joint. Int. J. Fatigue **105**, 276–282 (2017)
26. Djurdjevic, A., Zivojinovic, D., Grbovic, A., Sedmak, A., Rakin, M., Dascau, H., Kirin, S.: Numerical simulation of fatigue crack propagation in friction stir welded joint made of Al 2024-T351 alloy. Eng. Fail. Anal. **58**, 477–484 (2015)
27. Colic, K., Sedmak, A., Grbovic, A., Burzić, M., Hloch, S., Sedmak, S.: Numerical simulation of fatigue crack growth in hip implants. Proc. Eng. **149**, 229–235 (2016)
28. Belytschko, T., Black, T.: Elastic crack growth in finite elements with minimal remeshing. Int. J. Numer. Methods Eng. **45**(5), 601–620 (1999)
29. Moës, N., Dolbow, J., Belytschko, T.: A finite element method for crack growth without remeshing. Int. J. Numer. Methods Eng. **46**(1), 131–150 (1999)
30. Mohammadi, S.: Extended Finite Element Method for Fracture Analysis of Structure. Blackwell Publishing Ltd., Oxford (2008)
31. Abdelaziz, Y., Hamouine, A.: A survey of the extended finite element. Comput. Struct. **86**(11–12), 1141–1151 (2008)
32. Affatato, S., Colic, K., Hut, I., Mirjanić, D., Pelemis, S., Mitrovic, A.: Short History of Biomaterials Used in Hip Arthroplasty and Their Modern Evolution. Biomaterials in Clinical Practice. Springer, Cham (2018)
33. Sedmak, A., Čolić, K., Burzić, Z., Tadić, S.: Structural integrity assessment of hip implant made of cobalt-chromium multiphase alloy. Struct. Integr. Life **10**(2), 161–164 (2010)
34. Colic, K., Sedmak, A., Gubeljak, N., Burzic, M., Petronic, S.: Experimental analysis of fracture behavior of stainless steel used for biomedical applications. Struct. Integr. Life **12**(1), 59–63 (2012)
35. Cvijović-Alagić, I., Cvijović, Z., Mitrović, S., Panić, V., Rakin, M.: Wear and corrosion behaviour of Ti–13Nb–13Zr and Ti–6Al–4V alloys in simulated physiological solution. Corros. Sci. **53**(2), 796–808 (2011)
36. Hloch, S., Monka, P., Hvizdos, P., Jakubeczyova, D., Kozak, D.V., Colic, K.G., Kl'oc, J., Magurova, D.: Thermal manifestations and nanoindentation of bone cements for orthopaedic surgery. Thermal Sci. **18**(1), S251–S258 (2014)
37. Kühn, K.D.: Bone Cements: Up-To-Date Comparison of Physical and Mechanical Properties of Commercial Materials. Springer, New York (2000)
38. Hernandez-Rodriguez, M.A.L., Ortega-Saenz, J.A., Contreras-Hernandez, G.R.: Failure analysis of a total hip prosthesis implanted in active patient. J. Mech. Behav. Biomed. Mater. **3**, 619–622 (2010)
39. Legweel, K., Sedmak, A., Čolić, K., Burzić, Z., Gubeljak, L.: Elastic-plastic fracture behaviour of multiphase alloy MP35N. Struct. Integr. Life **15**(3), 163–166 (2015)

40. Lee, E.W., Kim, H.T.: Early fatigue failures of cemented, forged, cobalt-chromium femoral stems at the neck-shoulder junction. J. Arthroplast. **16**(2), 236–238 (2001)
41. Woolson, S., Milbauer, J., Bobyn, J.D., Maloney, W.J.: Fatigue fracture of a forged cobalt-chromium-molybdenum femoral component inserted with cement: a report of ten cases. J. Bone Jt. Surg. Am. **79**(12), 1842–1848 (1997)
42. Lam, L.O., Stoffel, K., Kop, A., Swarts, E.: Catastrophic failure of 4 cobalt-alloy Omnifit hip arthroplasty femoral components. Acta Orthop. **79**(1), 18–21 (2008)