ISSN 1812-5123. Russian Journal of Biomechanics, 2018. Vol. 22, No. 4: 382-400

DOI: 10.15593/RJBiomech/2018.4.02

FINITE ELEMENT STRESS ANALYSIS OF A TOTAL HIP REPLACEMENT IN TWO-LEGGED STANDING

A.I. Borovkov[1], L.B. Maslov[1, 3], M.A. Zhmaylo[1], I.A. Zelinskiy[1], I.B. Voinov[1], I.A. Keresten[2], D.V. Mamchits[2], R.M. Tikhilov[4, 5], A.N. Kovalenko[4], S.S. Bilyk[4], A.O. Denisov[4]

[1] Institute of Advanced Manufacturing Technologies, Peter the Great St. Petersburg Polytechnic University, 29 Politekhnicheskaya Street, Saint-Petersburg, 195251, Russia e-mail: iamt@spbstu.ru
[2] Institute of Applied Mathematics and Mechanics, Peter the Great St. Petersburg Polytechnic University, 29 Politekhnicheskaya Street, Saint-Petersburg, 195251, Russia e-mail: office@iamm.spbstu.ru
[3] Department of Theoretical and Applied Mechanics, Ivanovo State Power Engineering University named after V.I. Lenin, 34 Rabfakovskaya Street, Ivanovo, 153003, Russia, e-mail: maslov@tipm.ispu.ru
[4] Russian Research Institute of Traumatology and Orthopedics named after R.R. Vreden, 8 Akademika Boykova Street, Saint-Petersburg, 195427, e-mail: info@rniito.org
[5] North-Western State Medical University named after I.I. Mechnikov, 47 Piskarevsky Prospect, Saint-Petersburg, 191015, e-mail: rectorat@szgmu.ru

Abstract. This paper presents a stress analysis of the biomechanical system formed by the hip joint prosthesis and pelvic bones with loading corresponding to a balanced two-legged stance. To this end, the geometric model of a previously designed implant is adjusted, a finite element model of the system is built, and a series of strength calculations in the “upright standing” position is performed. The model takes into account the porous structure of the prosthesis replacing the joint fragment lost due to physiological processes or trauma. Characteristics of computer analysis of revision hip replacement of the acetabular joint component are considered. The results of finiteelement analysis of stress-strain state of the system formed by the skeleton and hip prosthesis in two-legged stance are described. The main focus is on the calculation of the stresses in the pelvic component of the prosthesis under static loading produced by the tightening torque of screwsinserted into bone and the patient’s weight.

Key words: hip joint, prosthesis, finite element analysis, stress-strain state, revision arthroplasty.

© Borovkov A.I., Maslov L.B., Zhmaylo M.A., Zelinskiy I.A., Voinov I.B., Keresten I.A., Mamchits D.V., Tikhilov R.M., Kovalenko A.N., Bilyk S.S., Denisov A.O., 2018
Alexey I. Borovkov, Ph.D., Professor, Vice-Rector for Innovative Projects, Saint-Petersburg
Leonid B. Maslov, Ph.D., (Physics and Mathematics), Head of Department, Ivanovo, Senior Research, Saint-Petersburg
Mikhail A. Zhmaylo, Ph.D., Head Engineer, Saint-Petersburg
Ilya A. Zelinskiy, Ph.D., Engineer, Saint-Petersburg
Igor B. Voinov, Ph.D., Head of Department, Saint-Petersburg
Ilya A. Keresten, Ph.D., Engineer, Saint-Petersburg
Dmitry V. Mamchits, Engineer, Saint-Petersburg
Rashid M. Tikhilov, Ph.D., Professor, DMSc, Director, Saint-Petersburg
Anton N. Kovalenko, Ph.D., Senior Research Fellow, Saint-Petersburg
Stanislav S. Bilyk, Ph.D., Orthopedic Traumatologist, Saint-Petersburg
Alexey O. Denisov, Ph.D., Orthopedic Traumatologist, Saint-Petersburg

INTRODUCTION

Hip replacement is one of the most common surgical procedures and a fundamental issue in orthopedic biomechanics. The total hip prosthesis consists of an acetabular component that replaces the acetabulum and a femoral component that replaces the femoral head (Fig. 1).

The acetabular component is typically a metal cup with a polyethylene or ceramic liner. During the total hip replacement surgery, the cup may be press fitted into place, cemented, or screwed into an artificial acetabular socket. The femoral component of the hip prosthesis consists of a stem and a ball.

The prosthetic ball may be made of titanium or cobalt - chrome with ceramic coating. The most popular material for the prosthetic stem is titanium. The combinations of materials typically used for femoral head and acetabular cup are ceramic-on-ceramic, ceramic-on-polyethylene, and metal-on-polyethylene. The main medical goal of the hip replacement surgery is to restore the normal function of the joint.

The service life of the prosthesis can be reliable predicted only when taking into account such key characteristics of the task as the complex geometric shape of areas of the biomechanical system, its multiple components, dynamically heterogeneous stress conditions, and numerous contact interactions. One of effective solutions for this problem is direct finite element modelling. The need to take into account numerous details of the computational domain results in a multimillion system of mathematical equations, which can only be handled by supercomputer technologies. The need to consider anatomic limitations of a particular patient necessitates the use of additive technology.

Finite element analysis as a computational method for studying complex biological systems on the basis of continuum mechanics models has been actively used since the 1970s [11, 22]. At present, there are three main areas of finite element modelling in orthopedic biomechanics: 1) construction design and preclinical study of the strength of prostheses and various fixing devices used in osteosynthesis [21]; 2) theoretical analysis of core mechanical properties of the human musculoskeletal system; 3) investigation of complex time-dependent processes of adaptation, growth, and regeneration of live tissues using continuum mechanics methods [19].

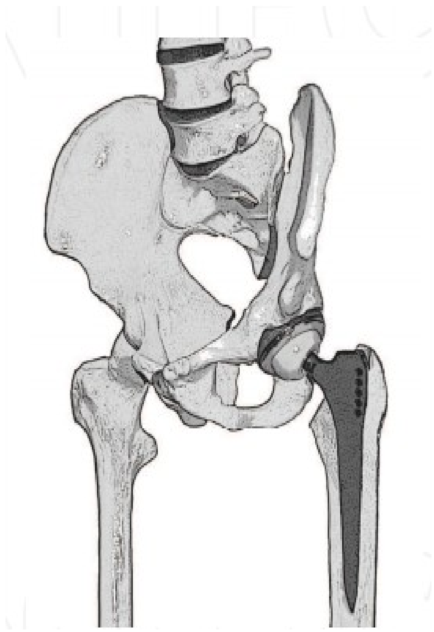


Fig. 1. Total hip replacement graph [12]

Since the end of the 20th century, there has been published a large number of papers dedicated to theoretical study of mechanical aspects of interaction between hip prosthesis and bone tissue using software systems based on the finite element method. The bulk of research focuses on the stress-strain state of the femoral and acetabular components of the hip prosthesis and bone tissue in primary hip replacement [1, 2, 15, 25, 27]. A great deal of literature is on the stress shielding effect near implants characterized by higher rigidity compared to bone tissue, and development of new models of prostheses with bone-like mechanical properties [3, 6, 8, 16].

The prosthesis is assumed to be formed by a standard set of components of a specific size and shape that are used in primary hip replacement surgery. As a rule, it is assumed that there is full mechanical contact between the prosthesis components and bone surface, while a number or papers specifically discuss relative micro - migrations of the prosthesis stem in the tibial canal. Modern approaches to stress analysis of hip replacement are typically based on computed tomography (CT) and geometric CAD (Computer Aided Design) implant models [13, 15].

The past decade, publications appearedthat describe the results of biochemical studies of the skeleton - prosthesis systems in revision hip replacement associated with significantly invasive surgery and customized implant selection. Thus, works [23, 30] discuss acetabular socket reconstruction with the help of implants produced using autologous biological materials harvested from the fibula. Papers [10, 29] consider the options for fixing the implant to the pelvis from the biomechanical point of view. Paper [12] presents a comprehensive finite element pelvis model including the lumbosacral spine and proximal femur based on the patient's CT scan and spatial geometry of the studied area. Finite elements of the mesh of the pelvic bone, elements and individual components of the hip prosthesis with complex spatial shapes are formed by solid elastic elements and assembled. A 500 N vertical load is applied to the fifth lumbar vertebra, and the pelvis is fixed so as to simulate upright bipedal stance. Calculations revealed a considerable concentration of stresses on the surface of the transplant made of fibular biomaterial. We suggest that peak stresses could be minimized by introducing additional internal fixation to transfer effort from the bone augment to the implant screw system. A high concentration of stresses was also found at the places where screws attached the acetabular component to the bone. Among the four methods of fixation, the best was double rod systems with stems and iliac screws, which yielded the lowest values of maximum stresses and the least pelvis migration.

Paper [31] presents a modular prosthesis that features customizable shape and size used for reconstruction of the lost half of the pelvis. The work offers a comparative analysis of stress distribution between the normal pelvis and restored pelvis at three static positions: sitting, standing on two legs, and standing on the leg on the affected side. Loads, the points of their application, and kinematic constraints on the degrees of freedom in this model are similar to the ones described in [12]. It is demonstrated that, in a normal pelvis, stress distribution is concentrated on the superior area of the acetabulum, arcuate line of the ilium, sacroiliac joint, sacral midline, and the superior area of the greater sciatic notch. In the restored part of the pelvis, stress distribution was concentrated in proximal area of the pubic plate, superior area of the acetabular socket, and the connection between the customized implant with the pelvic wing and sacroiliac joint. Overall, stress distributions in the postoperative and normal pelvis were similar in all three static positions. It is concluded that pelvic reconstruction with a modular hemiprosthesis had good biomechanical characteristics.

Paper [28] describes a novel approach to implant design based on 3D printing technology and additive layer manufacturing (ALM). The method is perfect for designing anatomically conforming custom-made implants with internal porous structures that improve osseointegration at the bone - implant junctures. This technology was described in a study of total hip arthroplasty with extensive acetabular bone loss and pelvic discontinuity [7]. Patient-specific acetabular implants are first designed to ensure precise correspondence to the affected areas of the bone, and then manufactured using a 3D printer.

Despite the recent achievements in hip replacement surgery, there remain a range of challenges that require further investigation of implant construction and materials, materials of the contacting bodies and many other issues. The finite element method is used in implant design and helps to answer unresolved questions related to clinical difficulties.

Our primary objective was to perform strength testing of the acetabular component of a customized prosthesis for revision hip arthroplasty. Unlike in primary hip replacement, patients undergoing revision surgery have individual bone defects, which means that the standard solution, whereby prostheses are simply selected from a catalogue, may fail. Revision surgery will be have maximum chances of success and will ensure the durability of the construction if the prosthesis fairly precisely restores joint geometry and the mechanics of the patient's movements. This is particularly important in revision surgery as every operation affects the quality of the bone and each successive surgery will increase in complexity. This is what requires the use of customized implants designed with the help of computer engineering technology and additive manufacturing, which together allow achieving maximum precision and customization.

This paper, which is based on a presentation [14], discusses the specific tasks of computer modelling of acetabular component revision in hip replacement. It presents results of finite element stress analysis of the system formed by the skeleton and hip prosthesis during two-legged standing. The main focus is on the calculation of the stresses in the pelvic component of the prosthesis under static loading produced by the tightening torque of screws inserted into bone and the patient's weight.

MATERIALS AND METHODS

Objective

A total replacement of the patient's right hip joint by an artificial endoprosthesis was performed at Russian Scientific Research Institute of Traumatology and Orthopedics named after R.R. Vreden (St. Petersburg). As indicated clinically, a portion of the ilium, which forms part of the pelvic bone system, was removed along with the acetabular socket, which acts as the hip-femur juncture. As a considerable part of the ilium was removed, a large titanium implant was put in its place, with porous coating to ensure effective osseointegration over time. Initial implant fixation was performed using a titanium plate and medical screws to secure the prosthesis in the pelvis. The plastic cup (liner) of the prosthesis was cemented to the artificial acetabular socket created using a titanium implant, ensuring that the cup and bone surfaces fit snugly together. Relatively standard surgical procedures to put in place the femoral component were also performed.

During the preoperative clinical assessment, we used CT and standard software to generate 3D geometric models of the hip system formed by the sacrum, femoral heads, and left and right pelvic bones including ilia and acetabular sockets.

Because the goal was a detailed stress analysis of the acetabular component of the prosthesis at the juncture with the ilium, no 3D models of the femur were designed. The goal of computer modelling of the two-legged stance is shown in Fig. 2.

The femoral component of the prosthesis is shown relatively roughly as a simple cross-section rod in order to retain the characteristic sizes of the entire skeleton - prosthesis biomechanical system.Given symmetrical loading from the body weight and working muscles, for a significant reduction in calculation time, it is reasonable to consider one half of

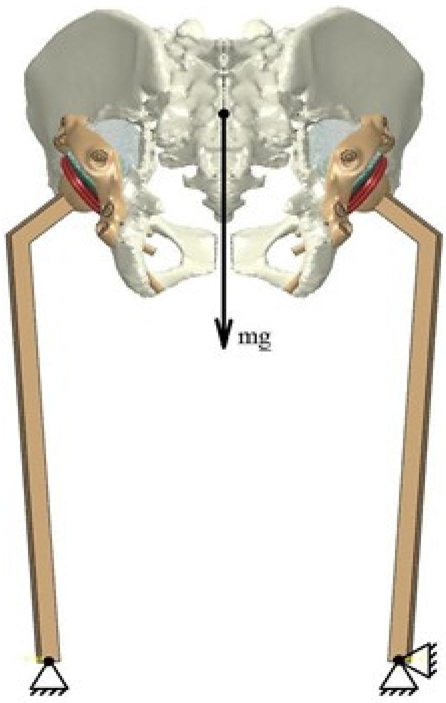


Fig. 2. Formulation of the problem for calculation of strength of the skeleton - hip prosthesis system

the computational domain, even though in reality only one half of the hip joint was replaced with a prosthesis. Therefore, in Fig. 2 the left side of the system was obtained by symmetrically mirroring the right side. It is assumed that replacing the hip joint with an artificial implant should ensure proper functioning of the human musculoskeletal apparatus and has been performed without making considerable changes to the general distribution and effects of the forces from the main motor muscles.

Finite element models

Geometric spatial models of the sacral and pelvic bones were processed, positioned and revised using the Solid Works software system for automated design (Dassault Systèmes, USA) [14].

The design of spatial finite element meshes based on 3D geometric models, development of the full computational model, configuration and solution were performed using the ABAQUS CAE computer engineering software system (Dassault Systèmes, USA).

The computational model consists of three structural groups: the pelvis and sacrum, acetabular component, and femoral component. The main groups of components of the finite element model simulating the skeleton - prosthesis biomechanical system are shown in Figs. 3, 4.

The pelvis is assumed to be formed by a 0.5 mm external layer of compact bone (cortical bone) and spongy substance that fills in the remaining internal volume (trabecularbone) (see Fig. 3). This group also includes the half of the sacrum tightly attached to the bulk of the pelvis.

The femoral component group comprises the titanium femoral component of the prosthesis that replaces part of the ilium, a plastic cup, and bone cement holding the liner in the implant (see Fig. 4). As mentioned above, the model of the femoral bone with the

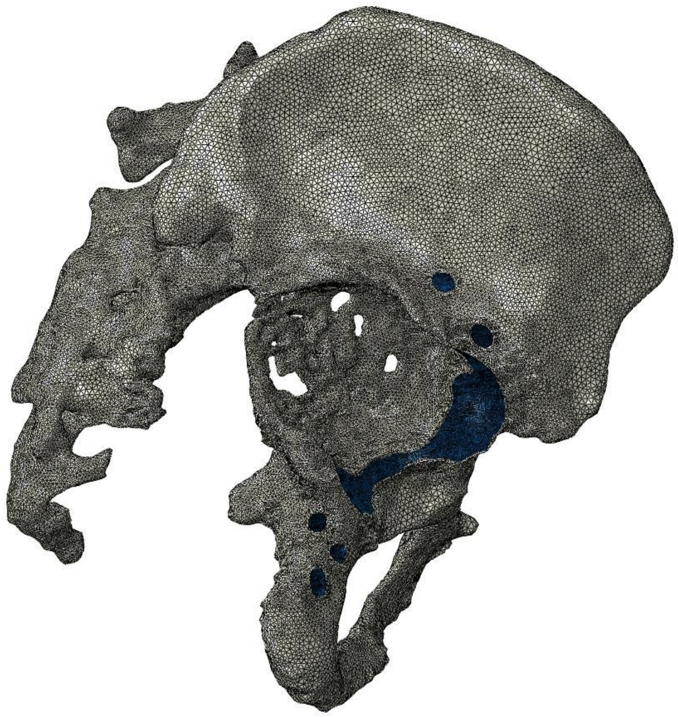
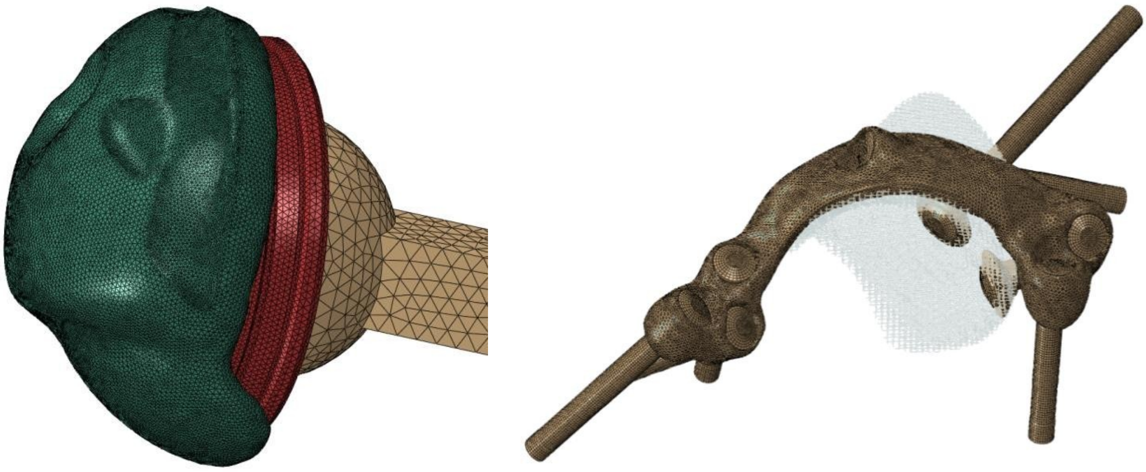


Fig. 3. Right pelvis and sacrum group



a *b*

Fig. 4. Prosthetic components group: femoral component group (*a*), acetabular component group (*b*)

prosthesis isnot considered in this formulation of the problem, and the femoral component is a modelled as a simple construct formed by a 3D stem and ball corresponding to the inner diameter of the implant cup.

Finally, the acetabular component may be considered to include an all-metal titanium plate, porous titanium augment, and special screws used to fix the metal components to thehealthy part of the pelvic bone (see Fig. 4).

Frontal and lateral views of the complete finite element model of the skeleton - prosthesis system are shown in Fig. 5. The principal element used for the majority of 3D parts of the system is the linearly elastic tetrahedral finite element with three translational degrees of freedom in the node.

a *b*

Fig. 5. Computational model of the skeleton-prosthesis biomechanical system: front view (*a*), right view (*b*)

The exception is the porous titanium augment replacing part of the ilium in the area of the acetabular socket. Because this detail was made of titanium with a considerably degree of porosity (70%), its finite element model was built using beam elements with translational and rotational degrees of freedom, forming a spatial construct from basic cells.

Quantitative characteristics of the finite element mesh of the biomechanical system components have the following values:

- pelvic bone: 1 080 000 elements, 500 000 nodes;
- titanium implant: 800,000 elements, 160 000 nodes;
- bone cement: 360 000 elements, 75 000 nodes;
- liner (cup): 210 000 elements, 40 000 nodes;
- titanium plate: 180 000 elements, 40 000 nodes.

The total number of finite elements in the skeleton - prosthesis computational model was 2 800 000, while the total number of degrees of freedom, which determines the size of the global stiffness matrix, was approximately 6 060 000. The computations were done on a work station with a 6th generation Intel i7 processor, 32 GB RAM.

Characteristics of materials

Hooke's body was taken as the main mathematical model for materials used in the finite element components of the biomechanical construction. Despite the presence of bone tissue microstructure and porous titanium augment, all the materials were regarded as homogenous, with isotropic effective properties.

The review of literature on the values of density, elasticity module, Poisson's ratio, stress limits and the coefficient of friction in contact pairs showed a fairly large range even for standard materials such as titanium, polyethylene and bone cement (Tables 1, 2). Therefore, as a rule, average values from the value range were chosen for calculations (bold in Tables 1, 2).

Table 1

Physical and mechanical properties of materials

Material	Elastic modulus, GPa	Critical stress, MPa	Poisson's ratio	Density, g/cm^3
Ti6Al4V Titanium (all metal parts)	105-**110**-115 [4]	1020-**1065**-1110 [4]	**0.3**-0.34 [4, 20]	4.4-**4.41** [4, 20]
Compact bone (cortical bone)	9.6-**10**-17.4 [26, 20, 5]	80-**100.5**-121 [20]	0.2-**0.3**-0.32 [26, 20, 5]	1.0-**1.47**-1.94 [20]
Spongy substance (trabecular bone)	**0.5**-1.5 [26, 5] 4.45-14 [21]	48-**53.5**-59 [20]	**0.3** [26, 20, 5]	**0.5**-1.19-1.87 [20]
Bone cement	**2.177** [24] 23.9-26.5 [20]	**95.9** [24]	**0.4** [20]	1.57 **-1.785** -2 [20]
Polyethylene (acetabular cup liner)	**25** [26]	12-**23.5**-35 [9]	**0.35** [26]	0.9-**0.93**-0.96 [9]

Table 2

Coefficients of friction for pairs of materials

Pair of materials	Coefficient of friction
Titanium (Ti6Al4V) - Titanium (Ti6Al4V)	0.3-**0.55**-0.8 [20], 0.15 [17]
Titanium (Ti6Al4V) - Cortical bone	0.1-**0.2** [20] - 0.3 [17]
Titanium (Ti6Al4V) - Trabecular bone	**0.3** [17]
Titanium (Ti6Al4V) - Polyethylene liner	0.1-**0.15**-0.2 [20]
Cortical bone - Bone cement	**0.5** [17]
Trabecular bone - Bone cement	**0.5** [17]

Loads and kinematic constraints

In the original formulation of the problem, a force equivalent to the patient's weight is applied at the center of gravity located in the human body plane of symmetry (Fig. 2). As shown by static equilibrium equations, in bipedal standing it is balanced by the reaction forces of the supports that may be considered as equal to half the patient's weight. From the computational point of view, it is more convenient to fix the sacral area considering the pelvis to be stationary, and apply the load to the area at the end of the simplified model of the femoral component. Therefore, the following types of constraints on the translational degrees of freedom in the nodes of the finite element mesh (Fig. 6) were chosen as kinematic boundary conditions for the computational model:

Fig. 6. Boundary conditions imposed on the model

1) Symmetry conditions in the form of prohibition of normal displacement in the nodes of elements at the hip bone plane of symmetry (blue-orange area in Fig. 6).

2) Rigid fixation of the area of the sacrum, which articulates with the pelvis, through setting zero displacements in the finite element nodes (orange area in Fig. 6).

3) Fixation of the mesh nodes on the lower surface of the prosthesis stem to prevent displacement in the lateral plane while retaining their ability to move in the vertical direction (red arrow in Fig. 6), which is necessary to be able to apply the force corresponding to the patient's weight.

In this finite element model of the skeleton - prosthesis biochemical system, force boundary conditions are formed by two types of loads applied.

First, this is the 650 N reaction force of the support surface, equivalent to the patient's weight (130 kg). It is shifted along the line of action so to be applied to the center of the lower surface of the femoral component stem model in the direction of allowed vertical degree of freedom.

Secondly, external loads include specific forces that arise with the tightening into the volume of the hip bone of medical screws that pull the titanium components of the implant (the large porous titanium augment and titanium plate) to the hip bone. As the precise value of torque with which the surgeon tightens the screws is typically unknown, the magnitude of tension forces for cortical screws with a thread diameter of 4.5 mm was taken as 500 N. This value is confirmed both by literature data [10] and our own experiments conducted to assess the maximum axial force corresponding to the beginning of bone destruction during the tightening of screws.

As the finite element model consists of several structural components that contact and interact with one another with varying intensities of constraints, contact interactions are introduced between corresponding surfaces. In this computational model, there are two types of contact pairs formed by contacting surfaces of spatial parts of the model depending on the degree of articulation between them.

The first group comprises the parts of the system that allow sliding and separation relative to one another, which is described by contact interaction with a possibility of relative displacements. This group comprises the following main contact pairs: 1) titanium head of the femoral component and polyethylene liner of the acetabular component; 2) porous titanium augment and pelvic bone; 3) titanium plate and pelvic bone; 4) screw heads and surfaces of the titanium plate. By including in the first group the porous titanium augment that creates an artificial acetabular socket, we meant that the performed computer calculations correspond to the initial stage after the hip replacement, when the bone has not yet grown into the pores of the prosthesis, and, consequently, micro - migration of the implant relative to the hip bone is still possible.

The second group comprises the parts of the system that, based on medical, biological and physical assumptions, must be rigidly attached to one another during the osteosynthesis surgery.

These include: 1) medical bone cement and the polyethylene liner of the acetabular component; 2) medical bone cement and the porous titanium augment; 3) titanium plate and porous titanium augment; 4) screw shafts and bone matter.

At the end of this section, it should also be noted that the model does not regard the hip bone as separate from the sacrum. As any contact pair has a major effect on the convergence rate of the iterative solution, to reduce the calculation time and considering the sacrum's remoteness from the area of interest, it was assumed that the volumes of the sacrum and the hip bone could be joined into a single body. The calculations were carried out in two stages with consecutive application of the tightening forces of the screws and the patient's weight. Assumptions were made about small deformations that arise in the elements of the construct and the absence of stress relaxation effects for all the types of materials included in the model.

RESULTS AND DISCUSSION

Modal analysis

Modal analysis is usually carried out to determine the natural frequencies and the corresponding vibration modes, but it can also be used as a starting point for solving non-stationary problems. A correct numerical model should have no zero-frequency modes in modal analysis. This prerequisite guarantees that all of the contact interactions will be accounted for and there will be no disconnected elements in the model.

This paper examines the static problem of two-legged standing, and consequently, rather than employing modal analysis to investigate the mode frequencies and shapes of the skeleton - prosthesis system, we used it to verify the finite element model and, in particular, to check the workability of contact interactions and kinematic boundary conditions.

Fig. 7 presents the first mode shape corresponding to the lowest frequency of free vibrations equal to 178 Hz.

Analysis of the mode shapes shows that the finite element model behaves correctly as a single construct, performing free vibrations with selected constraints at natural frequency lying within acceptable number range [18].

Stress analysis of the prosthesis model

This section provides the results of stress computations and assesses the strength of the biomechanical construct elements comprising the main components of the artificial hip joints, taking into account both types of loading: tightening forces of the screws and the patient's weight.

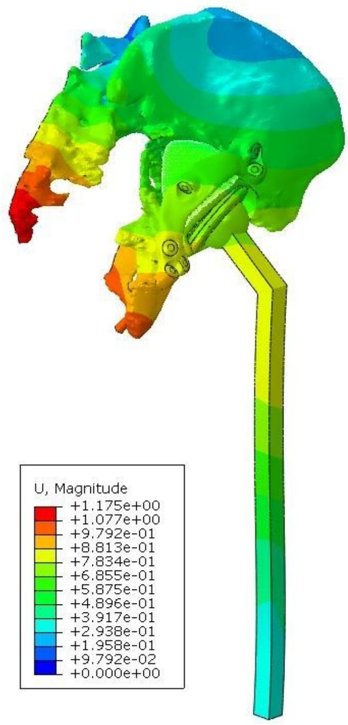


Fig. 7. Mode shape corresponding to the first natural frequency (178 Hz)

Fig. 8 shows the von Mises stress intensity distribution for the titanium femoral component of the prosthesis. The stresses are concentrated on the surface of the internal angle of the prosthesis stem at the juncture of the oblique and vertical parts of the prosthesis. Maximum stresses reach 148.2 MPa, which is considerably lower than the titanium ultimate tensile strength of 1065 MPa (Table 1). As could be expected, the simplified model of the femoral stem designed without considering the real shape and sizes of the stem fulfill strength requirements with a large margin because the section size of the rod simulating the stem was chosen a priori larger than the actual values.

The von Mises stress intensity distribution for the polyethylene liner is shown in Fig. 9. The largest concentration of maximum equivalent stresses is found in the area of edge contact interaction.

Stress plashes in individual points are irregular and localized, which is determined by singular edges and, possibly, weak finite element approximation of the computational domain. By neglecting these 3 plashes reaching up to 59.8 MPa and most likely arising through numeric errors, it may be concluded that, in accordance with Table 1, the liner retains its strength. A more precise stress analysis is possible if the prosthetic cup and the acetabular component are regarded as a separate construct, and if computations are made with a more detailed finite element mesh near the edge of the liner. In addition, it may be needed to flatten the right angle between the surfaces and take into account the elastic and plastic behavior of the material.

The distribution of equivalent stresses that arise in the bone cement model is also markedly localized (Fig. 10). Maximum stresses reaching 37.4 MPa are concentrated in the narrow area where the cement attached to the pelvic bone surface, and also between the contacting areas of the cement and titanium implant. However, in general it can be seen that bone cement almost does not experience considerable mechanic stress. In accordance with Table 1, the strength requirements for the bone cement are fulfilled with a safety factor >2.5.

The next group of components in which stress analysis is vital to assess their strength is the titanium plate and porous titanium augment.

Fig. 11 shows the von Mises stress intensity distribution for the plate used for initial fixation of the implant. The maximum stresses are largely concentrated near the screw holes and in the area of contact interaction with the bone caused by the local strain of titanium screws. Strength requirements are fulfilled with a significant safety factor >3.5 (Table 1). Such low values of equivalent stresses and the general stress distribution pattern suggest that the construct is suboptimal in terms of geometric and mass characteristics.

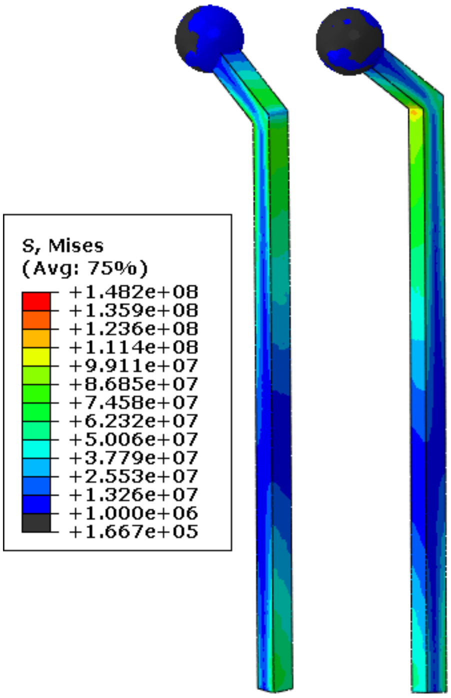


Fig. 8. Von Mises stress intensity for the femoral component

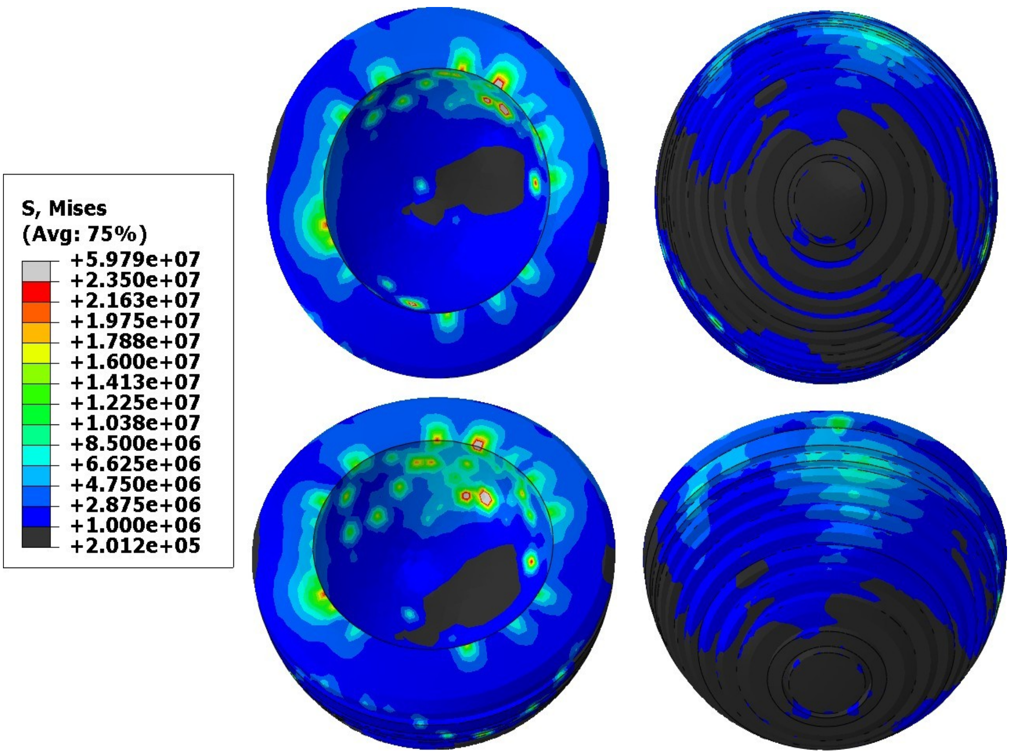


Fig. 9. Von Mises stress intensity for the polyethylene liner

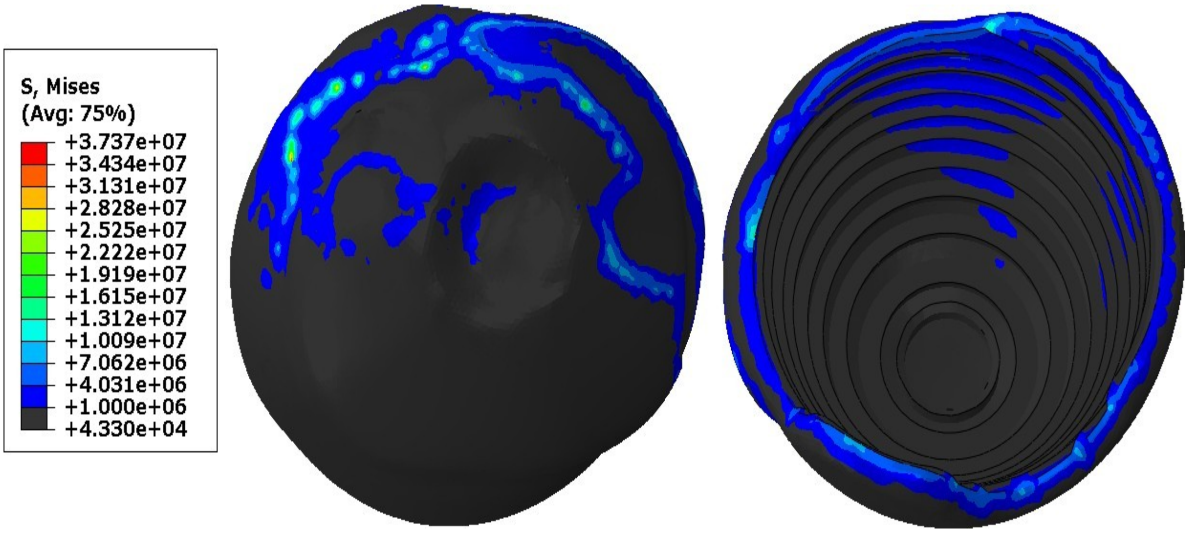


Fig. 10. Von Mises stress intensity for bone cement

When considering possible optimization, of course, it is important to note that the paper presents only results of static analysis, while dynamic loads may be several times larger. However, low stresses over the large part of the plate shows that the construct allows reducing the mass and volume of the material.

Stresses in medical titanium screws prove to be approximately twice as high as in the plate with which they interact (Fig. 12). Von Mises equivalent stresses caused by local strain along the axis of elongation are chiefly found on the surface of the screws outside the pelvic bone. According to Table 1, the factor of safety is at least 1.6.

Fig. 13 shows the von Mises stress intensity for the porous titanium augment that replaced the removed part of the pelvis in the area of the acetabular socket.

For the model of porous material built from beam elements, maximum stress concentration is found in localized areas created by singular contact interaction between the rods of titanium porous structure and the hip bone.

Increased stress in certain bars should probably be regarded as numerical errors associated with the characteristics of modelling contact interaction between one-dimensional beam finite elements (titanium augment) and spatial elastic elements describing the volume of hip bones.

As an alternative, the titanium augment may be described using popular phenomenological models of the porous body as a material continuum [18], which are also computerized in the form of finite element complexes. In this case, the properties and stress-strain state resulting from contact interaction will be "spread out" throughout the volume of the material. Integral macroscopic loads will be described correctly, while information concerning the microbehavior of the materials, including the results of contact interaction, will be averaged out. It may prove necessary to design additional models taking into account the augment microstructure in order to obtain a more precise picture of the stress-strain state.

However, even when using the bar model, peak stresses rapidly fade when moving away from the points of contact. It may be concluded that the size of critical areas could be disregarded in accordance with Table 1 and assume the safety factor of approximately 3.

Stress analysis of the pelvis model

This section presents the results of calculations of stresses that arise in the cortical and trabecular layers of pelvic bones due to the tightening torque of screws inserted into bone and

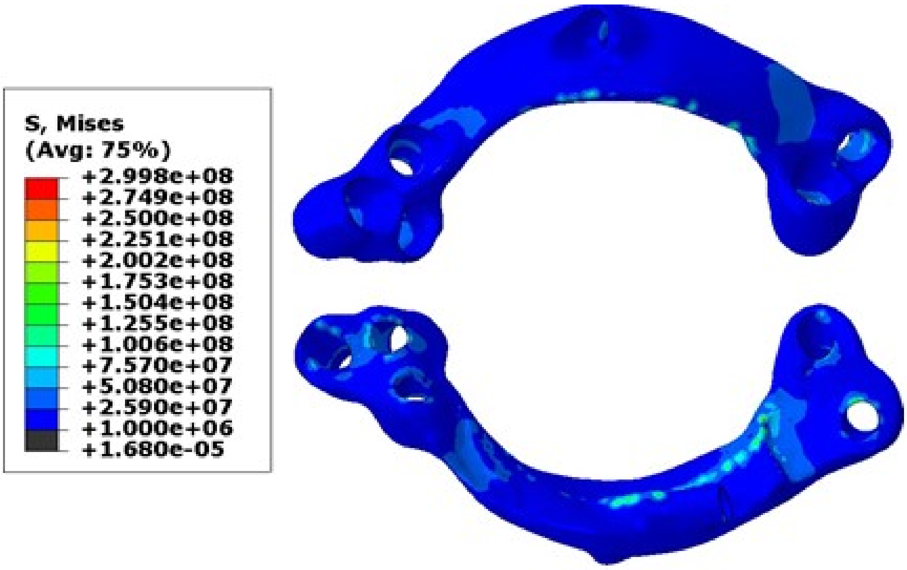


Fig. 11. Von Mises stress intensity for bone plate

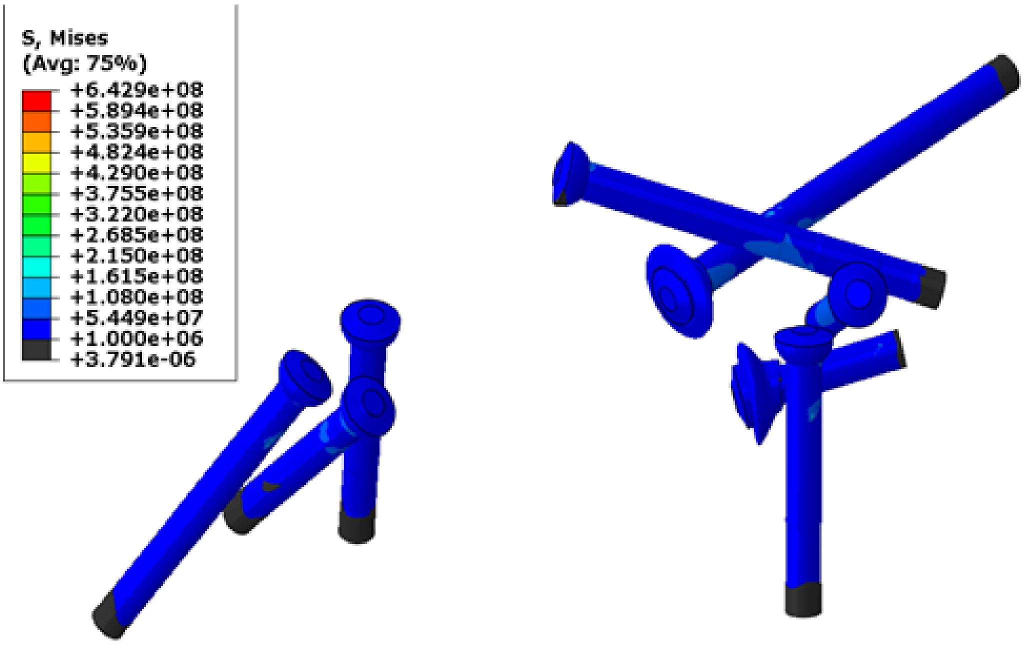


Fig. 12. Von Mises stress intensity for titanium screws

the patient's weight. Fig. 14 shows the distribution patterns of von Mises equivalent stress on the surface of the hip bone. Near-critical stresses are mainly concentrated in the cortical layer around the holes for the titanium screws; these are caused by the tension of the titanium screws securing the prosthesis to the ilium and other parts of the pelvis. The area of the acetabular socket bears an integrated load and is the most vulnerable. Calculations show that, in local areas near the edges of the screw holes, stresses exceed the safety factor of the cortical bone assumed to be 100 MPa.

From the practical point of view, this means that tightening the screws into the bone followed by loading the hip joint may result in damage of the adjacent layers of bone tissue.

The stress-strain state of the hip bone may be exacerbated by faults in the construct or geometric discrepancy of the contacting surfaces.

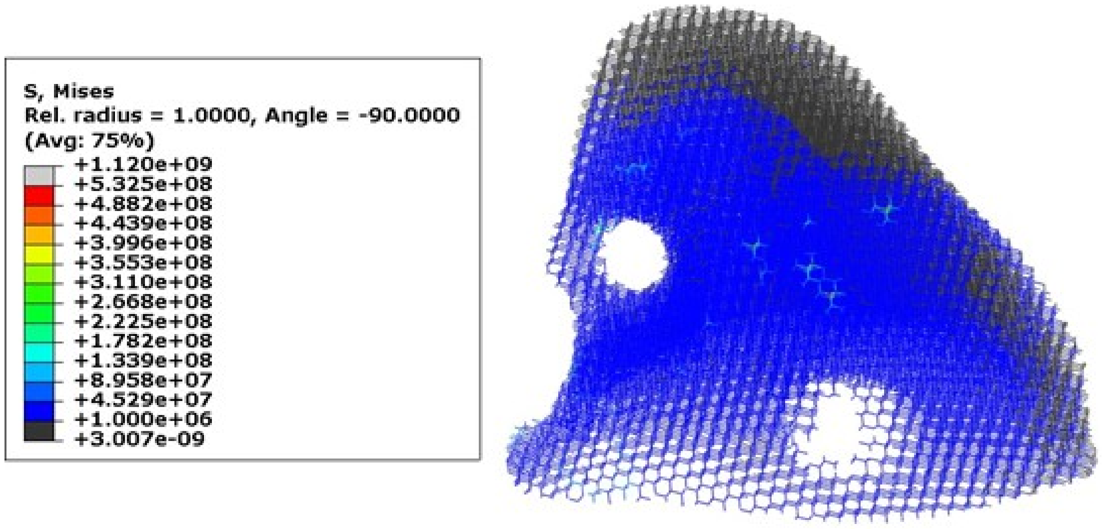


Fig. 13. Von Mises stress intensity for the porous titanium augment

Figs. 15, 16 show stress fields calculated after the first and second stages of loading, respectively, in order to demonstrate which forces lead to the concentration of stress in the bone tissue near screw holes. In accordance with the mechanical formulation of the problem, during the first stage the construct is loaded by the screw tightening forces with contact interaction, while the force equivalent to the patient's weight is applied at the next computational stage.

Comparative analysis of the calculations of the hip bone stress-strain state without (Fig. 15) and with (Fig. 16) the patient's weight respectively proves that the concentration of stresses at the edges of the holes is caused by the axial forces with which the screws pull the bone tissue to the surface of the titanium plate. Application of equivalent weight load has no effect on the stress distribution in the area of the screws and contact connection of the prosthesis with the bone, with the maximum stress value increasing only by 3%, which is not critical considering that the stress considerably exceeds, if only locally, the set safety factor.

At the same time, as can be expected, the body weight raises the general stress level in the entire volume of the pelvis, including the back surface of the ilium, inferior parts of the hip bone, and the sacrum. However, in these areas stresses do not exceed 50% of the cortical tissue safety factor as given in Table 1.

To reduce the stress levels in the bone and ensure short-term (before osseointegration is completed) reliable fixation of the prosthesis elements, we can propose the following recommendations that result from the identified functional characteristics of this biomechanical construct. Primarily, during the surgery, it is advisable to avoid the formation of thin walls between adjacent screws by spacing them out.

It is also advisable to avoid combinations of different stress concentrators, e.g. opening of ellipsoidal screw holes at the arcuate lines of the pelvis. The combination of such stress concentrators leads to a cumulative increase of the stress-strain state. When positioning is possible, the direction of the screw must be changed so that the hole does not open at the arcuate line of the pelvis.

Finally, for a more uniform distribution of contact pressure and, as a result, lower stress peaks on the surface of the hip bone near the screw holes, it is necessary to select anchoring elements with a maximum topological correspondence of working surfaces with the surface of the hip bone.

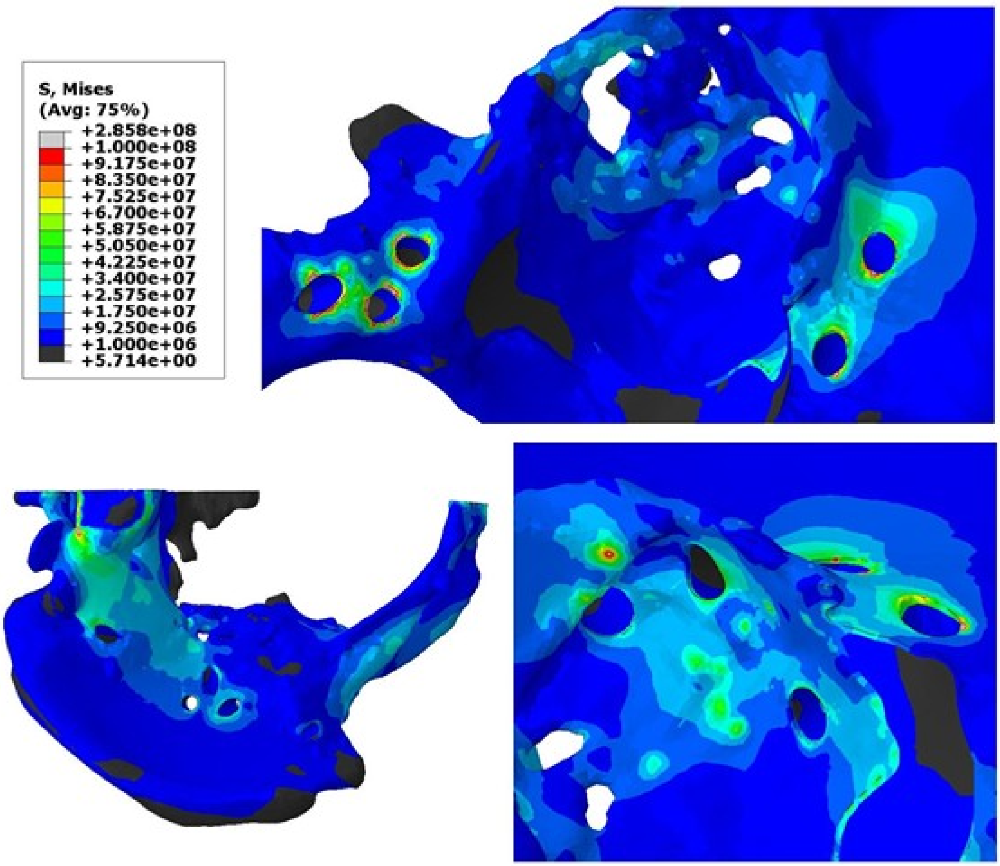
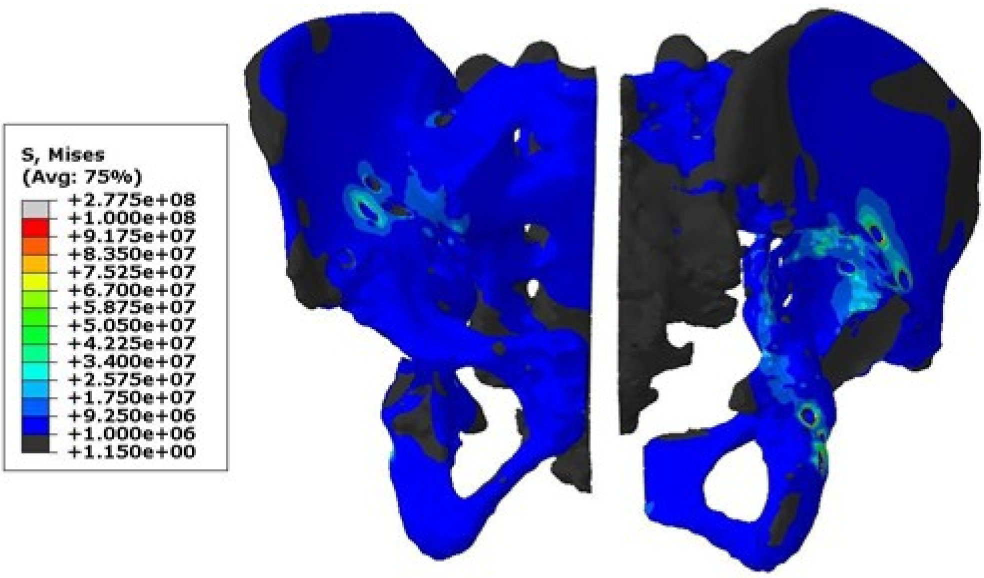
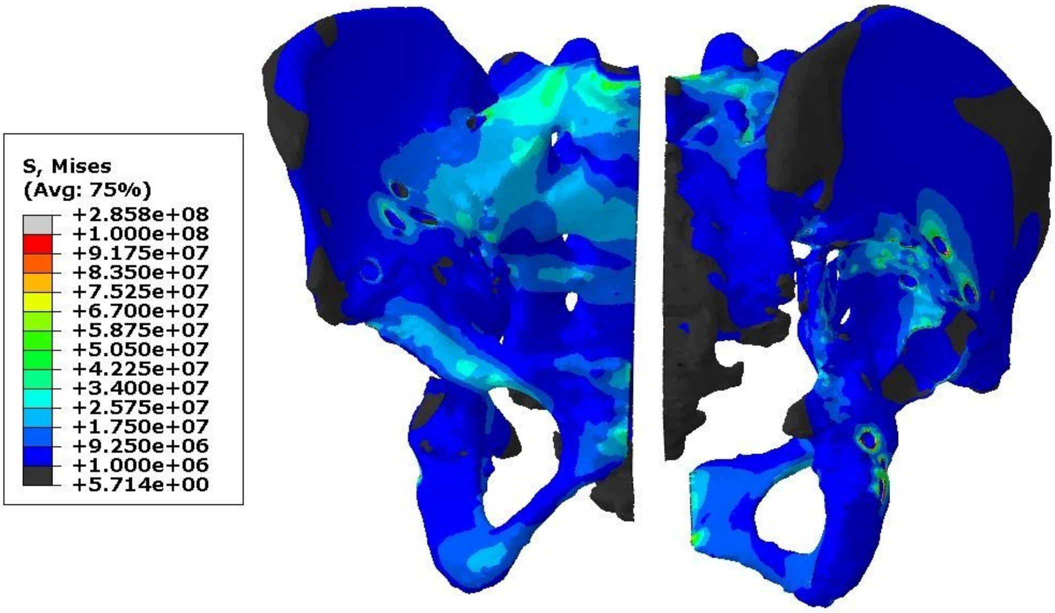


Fig. 14. Von Mises stress intensity for the hip bone



a

Fig. 15. Von Mises stress intensity for the hip bone without (*a*) and with (*b*) the patient's weight (front and back views)



b

Fig. 15. Ending

Conclusions

The paper examines an important issue in the biomechanics of the musculoskeletal system: a theoretical stress analysis of the skeleton - hip prosthesis system in a two-legged stance. Based on the patient's CT scan results, we developed realistic finite element models of the pelvic bones of the patient who underwent surgery at Russian Scientific Research Institute named after R.R. Vreden (St. Petersburg) and acetabular component of the patient-specific prosthesis.

This problem has become pressing with the development of personalized medicine that requires designing customized prostheses to restore lost functions of the elements of the human musculoskeletal system. Modern computational methods and software tools based on them allow us to solve contact problems of continuum mechanics that describe complex interactions between the bone tissue and implants in the presence of physiological loads and additional forces.

Computer analysis performed with the ABAQUS CAE universal finite element complex generate stress distribution patterns in the titanium elements of the prosthesis and the patient's pelvic bone tissues. The results and literature data on acceptable stress values were used to analyze the static strength of the skeleton - prosthesis biomechanical system.

Loading scheme of a two-legged stance consisted of two main stages: first - the tightening of the titanium screws; second - applying the patient's weight. It was established that the area most susceptible to damage is the cortical layer of the pelvic bone around the holes for titanium screws, which results from the local tension of the screws. It was shown that it is the first stage that gives rise to the principal and most significant concentration of the pelvic bone stress. Thus, from the point of view of ensuring pelvic bone strength, the surgery should focus on carefully attaching the prosthesis to the bone in order to avoid undesirable damage to the cortical bone layer around the holes.

Several recommendations may increase strength characteristics. First, it is important to monitor for signs of damage of the cortical layer of the hip bone when placing the prosthesis. The area of the cortical layer in the vicinity of the plate is subjected to two mechanical impacts: the tightening force of the screws and the pressure from the plate. Incomplete contact interaction between the plate - bone pair caused by partial mismatch of the geometric surfaces leads to a smaller contact surface and, as a result, higher stresses in this area, which is undesirable.

Second, it is recommended to determine the optimal spatial distribution of the fastening screws because positioning them too close together leads to the formation of thin walls that may adversely affect the integrity of the system. In addition, the combination of stress concentrators results in a considerably higher stress, which is undesirable for the strength and fatigue durability of the system. In particular, the ellipsoidal hole near the arcuate line of the pelvic bone is the most vulnerable area in terms of fatigue strength. By changing the direction of the screw hole by several degrees from the arcuate line so that the stress concentrators do not combine, it is possible to achieve a considerable improvement of strength characteristics.

It should be noted that, with time, the bone tissue will grow into the pores of the titanium implant [18], which will increase osteointegration of the porous titanium augment with the bone and a more uniform stress distribution from the patient's weight. In the long-term, the osteointegration effect should improve the fatigue strength assessment of the entire biomechanical construct.

References

1. Akulich Yu.V., Podgayets R.M., Scryabin V.L., Sotin A.V. The investigation of stresses and strains in the hip joint after operation of endoprosthetics. *Russian Journal of Biomechanics*, 2007, vol. 11, no.4, pp. 9-35.
2. Bedzinski R., Tyndyk M. A study of the strain and the stress distribution in the pelvis bone under different load condition. *12th Conference of the European Society of Biomechanics*, Dublin, 2000, pp. 335.
3. Boudeau N., Liksonov D., Barriere T., Maslov L., Gelin J.-C. Composite based on polyetheretherketone reinforced with carbon fibres, an alternative to conventional materials for femoral implant: Manufacturing process and resulting structural behavior. *Materials & Design*, 2012, vol. 40, no.9, pp. 148-156.
4. Boyer R., Welsch G., Collings E.W. Materials properties handbook: titanium alloys. *ASM International*, 1994. 1176 p.
5. Brown T.D., Ferguson A.B. Mechanical property distributions in the cancellous bone of the human proximal femur. *ActaOrthop. Scand*, 1980, vol. 51, no.1-6, pp. 429-437.
6. Cilla M., Checa S., Duda G.N. Strain shielding inspired re-design of proximal femoral stems for total hip arthroplasty. *J. Orthop. Res*, 2017, vol. 35, no.11, pp. 2534-2544.
7. Colen S., Harake R., De Haan J., Mulier M. A modified custom-made triflanged acetabular reconstruction ring (MCTARR) for revision hip arthroplasty with severe acetabular defects. *Acta Orthop. Belg*, 2013, vol. 79, pp. 71-75.
8. Denisov A.S., Nyashin Yu.I., Akulich Yu.V., Zmeev Yu.A., Osorgin Yu.K., Podgayets R.M., Scryabin V.L., Sotin A.V. Some aspects of application of carbon composite material in human hip joint prosthetics. *Russian Journal of Biomechanics*, 1997, vol. 1, no.1, pp. 12-24.
9. Grigoreva I.S., Mejlihova E.Z. Fizicheskie velichiny: spravochnik [Physical Values: Handbook]. Moscow: Energoatomizdat, 1991, pp. 122-123 (*In Russian*).
10. Guo W., Tang S., Dong S., Li X. Resection and reconstruction for tumors of iliac bone. *Chin. J. Surg*, 2006, vol. 44, pp. 813-816.
11. Huiskes R., Chao E.Y.S. A survey of finite element analysis in orthopaedic biomechanics: the first decade. *J. Biomech*, 1983, no.16. pp. 385-409.
12. Jia Y.-W., Cheng L.-M., Yu G.-R., Du C.-F., Yang Z.-Y., Yu Y., Ding Z.-Q. A finite element analysis of the pelvic reconstruction using fibular transplantation fixed with four different rod-screw systems after type I resection. *Chinese Medical Journal*, 2008, vol. 121, pp. 321-326.
13. Kluess D., Wieding J., Souffrant R., Mittelmeier W., Bader R. Finite element analysis in orthopaedic biomechanics. *Finite element analysis*, 2010, pp. 151-170.
14. Keresten I.A., Borovkov A.I., Zhmajlo M.A., Kovalenko A.N., Bilyk S.S. Konechno-elementnoe modelirovanie iissledovanie napryazhenno-deformirovannogo sostoyaniya komponentov tazobedrennogo sustava dly advuhopornogo stoyaniya [Finite element modeling and investigation of the stress-strain state of the components of the hip joint for two-support standing]. *Nedelya nauki SPbPU: materialy nauchnoj*

konferencii s mezhdunarodnym uchastiem. Institut prikladnoj matematiki i mekhaniki. Saint-Petersburg: Izd-vo Politekhn. un-ta, 2016, pp. 74-76 (*In Russian*).

15. Lerch M., Weigel N., Windhagen H., Ettinger M., Thorey F., Kurtz A., Stukenborg-Colsman C., Bouguecha A. Finite element model of a novel short stemmed total hip arthroplasty implant developed from cross sectional CT scans. Technol. *Health Care: J. Europ. Soc. Engineering and Medicine*, 2013, vol, 21, no.1, pp. 493-500.
16. Levadnyi I., Awrejcewicz J., Gubaua J.E., Pereira J.T. Numerical evaluation of bone remodelling and adaptation considering different hip prosthesis designs. *Clin. Biomech*, 2017, vol. 50, no.12, pp. 122-129.
17. Levadnyj E.V., Nushtaev D.V. Issledovanie napryazhenno-deformirovannogo sostoyaniya verhnej treti bedrennoj kosti prieemezhvertelnomperelomeina kostnoj fiksacii s primeneniem sheechnogo vinta so spiralevidnoj rezboj [Investigation of the stress-strain state of the upper third of the femur with its intertrochanteric fracture and bone fixation using a cervical screw with a spiral thread]. Moscow: MAKS Press, 2014. 168 p. (*In Russian*).
18. Maslov L.B., Liksonov D.V. Biomekhanicheskie harakteristiki nizhnej konechnosti cheloveka. Matematicheskie modeli i kompyuternoe modelirovanie v biomekhanike [Biomechanical characteristics of the human lower limb. Mathematical models and computer modeling in biomechanics: study guide]. Saint-Petersburg: Izd-vo Politekhn. un-ta, 2004, pp. 385-438 (*In Russian*).
19. Maslov L.B., Sabaneev N.A. Kompyuternoe modelirovanie biomekhanicheskih system kak sploshnyh sred so svojstvami strukturnoj perestrojki [Computer simulation of biomechanical systems as continuous media with structural adjustment properties]. Ivanovo: B.i., 2016. 176 p. (*In Russian*).
20. Nedoma J., Stehlik J., Hlavacek I., Danek J., Dostalova T., Preckova P. Mathematical and computational methods and algotithms in biomechanics of human skeletal systems: an introduction. John Wiley & Sons, 2011, 300 p.
21. Nemov A., Zhmaylo M., Zelinskiy I., Ermolenko N., Karandashev A., Borovkov A.,Khominets I., Mamedov K., Miliaev A., Grebenyuk A.. Simulation of mechanical behaviour of the plate for osteosynthesis: loads and computational approach. *Series on Biomechanics*, 2018, vol. 32, no. 1, pp. 30-37.
22. Prendergast P.J. Finite element models in tissue mechanics and orthopaedic implant design. *Clin. Biomech.*, 1997, vol. 12, no.6, pp. 343-366.
23. Sakuraba M., Kimata Y., Iida H., Beppu Y., Chuman H., Kawai A. Pelvic ring reconstruction with the double-barreled vascularized fibular free flap. *Plast. Reconstr. Surg*., 2005, vol. 116, pp. 1340-1345.
24. Spierings P., Breusch S., Malchau H. Properties of bone cement: testing and performance of bone cements. The well-cemented total hip arthroplasty. Springer, 2005, pp. 67-78.
25. Tanner K.E., Svensson I., Samuelsson F., Flivik G. Finite element study of the acetabulum in cemented hip arthroplasty investigating retention or removal of the subchondral bone plate. Biomedizinische Technik. *Biomedical Engineering*, 2016, vol. 61, no.5, pp. 525-536.
26. Tihilov R.M., Shubnyakov I.I., Mazurenko A.V., Mitryajkin V.I., Sachenkov O.A., Kuzin A.K., Denisov A.O., Pliev D.G., Boyarov A.A., Kovalenko A.N. Eksperimentalnoe obosnovanie ustanovki acetabulyarnogo komponenta s nedopokrytiem pri endoprotezirovanii pacientov s tyazheloj stepenyu displazii [Experimental substantiation of the installation of the acetabular component with the lack of coverage in endoprosthetics of patients with severe dysplasia]. *Travmatologiya i ortopediya Rossii*, 2013, no. 4, pp. 42-51 (*In Russian*).
27. Vasu R., Carter D.R., Harris W.H. Stress distributions in the acetabular region - I. Before and after total joint replacement. *J. Biomech.*, 1982, vol. 15, no.3, pp. 155-164.
28. Wong K.C., Kumta S.M., Geel N.V., Demol J. One-step reconstruction with a 3D-printed, biomechanically evaluated custom implant after complex pelvic tumor resection. *Computer Aided Surgery*, 2015, vol. 20, no.1, pp. 14-23.
29. Yamamoto Y., Takeda N., Sugihara T. Pelvic ring reconstruction with a vascularized bone flap of femur. *Plast. Reconstr. Surg*., 1997, vol. 100, pp. 415-417.
30. Yu G., Zhang F., Zhou J., Chang S., Cheng L., Jia Y. Microsurgical fibular flap for pelvic ring reconstruction after periacetabular tumor resection. *J. Reconstr. Microsurg.,* 2007, vol. 23, pp. 137-142.
31. Zhou Y., Min L., Liu Y., Shi R., Zhang W., Zhang H., Duan H., Tu C. Finite element analysis of the pelvis after modular hemipelvice endoprosthesis reconstruction. *Int. Orthopaedics*, 2013, vol. 37, pp. 653-658.

Received 15 September 2018