Application of reverse engineering for design of personalized hip implant

Zastosowanie inżynierii odwrotnej w projektowaniu spersonalizowanego implantu stawu biodrowego

JAROSŁAW ZUBRZYCKI
MAGDALENA BRANIEWSKA *

DOI: https://doi.org/10.17814/mechanik.2017.1.5

In the paper has been presented a methodology for designing personalized human hip prosthesis using Reverse Engineering methods based of the patient computed tomography (CT) studies. Are presented the results of numerical analyzes of strength tests received implants from FEM.

KEYWORDS: hip-joint implant, CAD, FEM, modeling, reverse engineering

Implantology and prosthetics of hip joint have been developing rapidly for over 40 years. Total or partial endoplasty is one of the most frequently performed procedures. The treatment is performed both in elderly people (aging-related injuries) and in young people (injuries). Implantation of a total prosthesis to a young person involves even several times the need for replacement. In order to increase the life expectancy of the prosthesis, new personalized construction solutions are created, innovative anatomical tests are carried out and increasingly improved biomaterials are produced.

It is important for the design and implementation of a personalized denture to adapt to the patient's anatomical dimensions. To meet these requirements, one should get the exact dimensions of the damaged joint. For this purpose, it is convenient to use a CT test, based on which a personalized prosthesis can be designed in CAD programs.

Biomechanics of the hip

A joint is a moving bone. The hip joint carries the greatest load, the articular bag and the ligament set can carry a load of up to 500 kg [2-4]. The hip joint is a spherical joint with three degrees of freedom in the transverse (flexion and straightening), sagittal (reversal and adaptive) and vertical plane (reversal and reversal) and one of the larger kinematic nodes in the human motion [3]. The structure of influences on the hip joint is very complex; it consists of a series of forces and moments resulting from body mass and muscle groups. Its function is also to convey the burden from the spine through the pelvis to the lower limbs during motion. A sudden load can lead to overdoing the bone strength and breaking its structure - fracture. Degenerative changes are caused by slow, long-lasting load. Among forces acting on the femur, forces within the joint (head of knee joint), muscular interactions, ligaments interactions, and inertia forces, can be distinguished.

As a result of the distance of the femoral head from the axis of the stem, an arm is created, which causes the bending moment. In the actual load system, the bone besides bending is also compressed and twisted.

The relation of loads occurring in the hip joint is presented in considerable simplification by means of a two-armed lever in which the support point is in the middle of the hip [1].

This analysis is approximately correct only if the center of gravity of a body is located in the frontal plane. Any movement causes the center of gravity to change, making changes in the directions and value of the forces coming from the muscles holding the body in equilibrium.

Modeling of the hip implant

Hip joint prosthesis can be selected for a particular patient based on computed tomography. The prosthesis was designed for a 23-year-old man weighing 85 kilograms and growing 180 centimeters after a mechanical hip fracture, directed to complete alloplasty. To adjust the implant, you need to select the geometric parameters that will ensure the correct anatomical biomechanics of the joint. The method of modeling and simulation analysis of the examined prosthesis includes:
- virtual reproduction of anatomical skeletal hip joint system based on CT,
- 3D model without cemented hip prosthesis,
- implant load analysis (MES).

For the analysis of the state of stresses and deformities induced in hip prosthesis, the model of hip joint load can be assumed according to Będziński [1]. It takes into account the influence of the body mass on the femoral head R, the effect of the muscle relaxant M_a, the response of the tibial bone band M, sliding on the trochanter T of rotators R_u and reaction from the surface R_p (fig. 1).

For the needs of the model, a simplified load scheme can be adopted, taking into account only the impact force resultant from the muscle to the hip joint. Force values F, muscle response M_a, T, M, and the ground response R_p are directly dependent on patient weight.

Anatomical bone and joint structures of the patient were mapped on the basis of CT imaging. A 64-line CT scanner was used - Philips Medical Systems, Ingenuity Core 128) and digital images of the lumbar spine were taken (fig. 2).

After segmentation of the tissues in the image, a spatial hip model was generated (fig. 3).

Models of fig. 3 were used to take the patient's anatomical dimensions. After export to CAD (CATIA), the model of the stem and the head of implant was made.

* Dr inż. Jarosław Zubrzycki (j.zubrzycki@pollub.pl) – Politechnika Lubelska, Wydział Mechaniczny, Zakład Inżynierii Biomedycznej; mgr inż. Magdalena Braniewska (braniewska.m@gmail.com) – Politechnika Lubelska, Wydział Mechaniczny

The numerical model of the implant was designed for MES strength analysis. Numerical tests used following materials:

- prosthesis – titanium alloy Ti6Al7Nb Protasul® 100, E = 115 GPa, v = 0.3;
- acetabulum - Al_2O_3 ceramics Biolux Al_2O_3-Keramik E = 410 GPa, v = 0.21-0.27.

A static load of 240 N, corresponding to the value in the heel-to-ground contact during normal walking (4.05 km/h) for a patient weighing 85 kg (F = 850 N) was applied onto the endoprosthesis. The numerical results are shown in fig. 4 and 5.

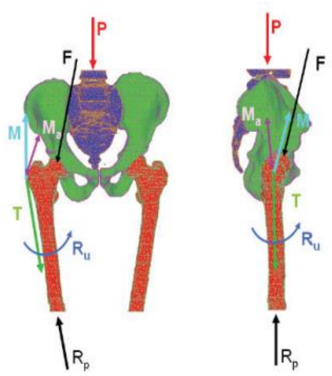

Fig. 1. General load model [1]

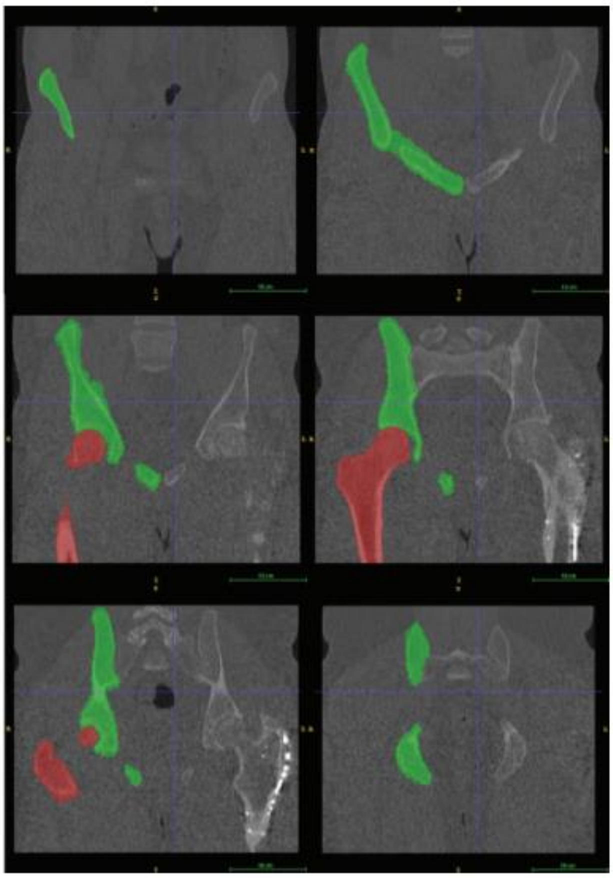

Fig. 2. CT examination of the spine along with segmentation of the hip joint

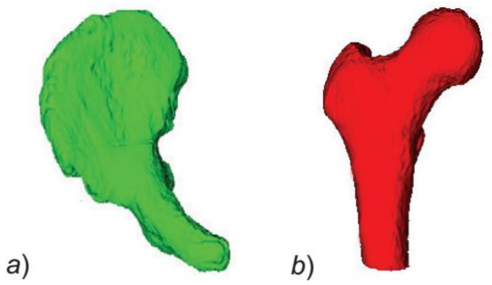

Fig. 3. Spatial model of pelvic bone (*a*) and femur (*b*) obtained from CT images

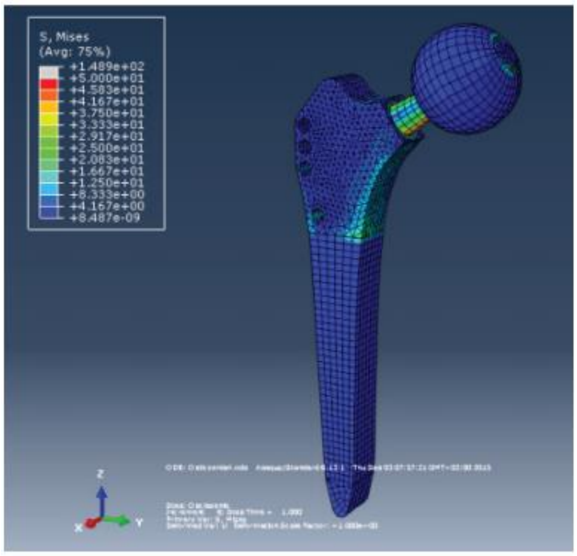

Fig. 4. Distribution of stresses reduced in the endoprosthesis

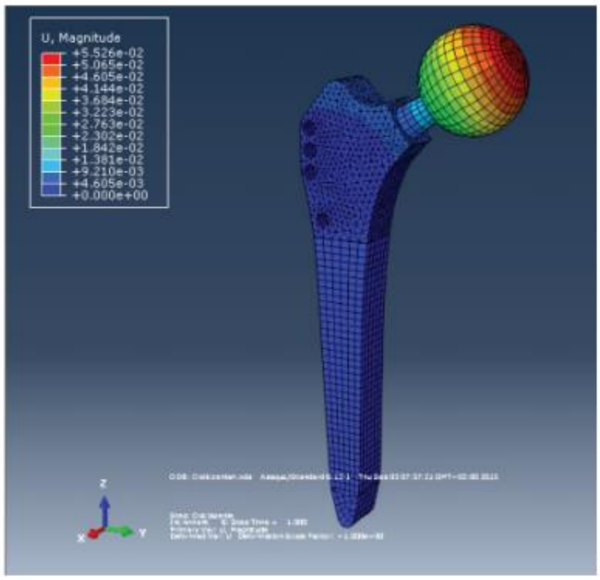

Fig. 5. Distribution of accidental displacement

Summary

Numerical results (fig. 4 and 5) show that the maximum stresses occur in the neck of the mandibular end of the prosthesis - they amounted to 149 MPa. In other parts of the prosthesis, the stress values ranged from 5 to 30 MPa. The main stresses were 32, 92 and 5 MPa respectively for directions S11, S22 and S12. The highest displacement values (head vault) were 0.05 mm.

From the results of numerical analysis, it can be concluded that these stresses are low enough and provide long life of the implant in the patient's body. Introducing the endoprosthesis to the body changes the biomechanics of the system and the load carrying capacity. When choosing the prosthesis, care must be taken to match the anatomical structure as closely as possible. Numerical analysis using MES enables determination and distribution of stresses reduced in the endoprosthesis as well as accidental displacement. This allows to estimate how the implant will behave in the bone structure. One can influence on the optimal selection of the prosthesis by choosing the appropriate shape and geometry of the mandrel, matching pairs of materials and their strength properties.

REFERENCES

1. Będziński R. „*Biomechanika inżynierska Zagadnienia wybrane*". Wrocław: Oficyna Wydawnicza Politechniki Wrocławskiej, 1997.
2. Reicher M. *Anatomia człowieka*. T. I: „Anatomia ogólna, kości, stawy i więzadła, mięśnie", 13th edition. Warszawa: Wydawnictwo PZWL, 2016.
3. Bernakiewicz M. „Koncepcja rozwiązania konstrukcyjnego trzpienia endoprotezy bezcementowej stawu biodrowego". *Biomechanika* 94. Prace Naukowe Instyt. Konstr. i Ekspl. Politechniki Wrocławskiej. Nr 75. Seria: Konferencja. Nr 21, Wrocław 1994.
4. Nam J., Choi H, Kang J. "Finite element analysis for friction noise of simplified hip-joint and its experimental validation". *J. of. Mechanical Science and Technology*. 30, 8 (August 2016): pages 3453–2460