

FINITE ELEMENT METHOD ANALYSIS OF HIP-JOINT PROSTHESIS

R. MONOSTORI✉, I. DUDÁS

University of Miskolc, Department of Production Engineering, 3515 Miskolc, Egyetemváros; MISKOLC
✉E-mail: hr2003@freemail.hu

Damage in the joints may entail deterioration of life quality to a great extent. More than seven hundred thousand big joint (hip-, knee, etc.) prostheses are implanted annually in the world to improve the quality of life, and this number in Hungary amounts to approximately ten thousand. Currently the lifetime of the prosthesis is approximately 10 years. Unfortunately, prosthesis loosens beyond this period, thus replacement is needed. In this paper we present the geometric construction, modelling and Finite Element Analysis of the hip-joint prosthesis.

Keywords: hip - joint prosthesis, 3D measurement, finite element method.

Introduction

Big joint prosthesis operations provide one of the most outstanding possibilities of locomotor surgery, and development is in progress. The hip joint prosthesis (*Fig. 1*) is one of the most complicated big joint prostheses, since it has to ensure a three-dimensional motion. Metal parts (for example, the stem and the sphere head) of hip prostheses are made of either stainless steel or another metal, for example, titanium. The companion piece (for example socket) is made of polythene [1]. The metal sphere head rotates in the polythene companion piece, which is inserted into the hip bone. The polythene has a number of favourable physical and chemical properties, for example, great strength, flexibility and chemical neutrality. However, the prosthesis made of this relatively ductile and resistant material, still wears.

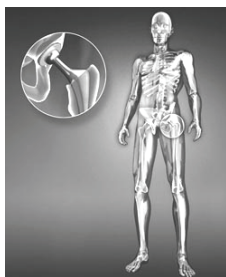


Figure 1: Hip joint prosthesis

Parts of the hip prosthesis

The anatomic hip prosthesis may be cemented and cementless depending on the type of fixation used to hold the implant in place. Construction:

- socket;
- metal head;
- metal stem of different types.

Basic Materials

The hip-joint stem and head are made of forged, heat treated, cobalt-based, patented special metal alloys, marked BMH-1, protected by a patent, and featuring a chemical composition in compliance with standard ISO 5832-6. The strength characteristics of the tissue-friendly material are the best. The high toughness (impact work), the special texture and chemical composition of the practically unbreakable metal result in the maximum resistance of the hip-joint stems manufactured from this material to the most unfavourable voltage corrosion fatiguing stresses.

The hip-joint socket is made of ultra-high molecular weight polyethylene in compliance with standard ISO 5834-2, and is applied successfully worldwide, on the basis of a procedure in which the material preserves its warranted characteristics.

Stem

The geometry of the stem (*Fig. 2*) is formed in a way that the curved contour line of the stem in the planar section of the plane of bending fully coincides with the cortical contour of the medullary space on the side

towards the centre of the body, this way, due to the large contact surface the applied load is transmitted evenly. This is favourable for the use of the bone. The wedging, straight stems are fixed in full length.



Figure 2: Stem [2]

Head

The hip-joint head is attached in a self-locked way to the conical neck part of the hip-joint stem (Fig. 3).

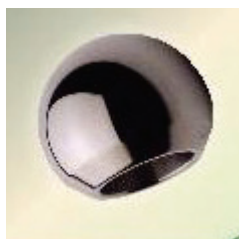


Figure 3: Head [2]

Socket

There are cams on the outer surface of the sphere to ensure the equal thickness of bone cement. In a special case a serrated-edge version of asymmetric arrangement is also available to increase the support function. The socket is fitted with marker wires at the right angles to enable X-ray check-ups after implantation (Fig. 4). The use of bone cement is necessary for the implantation.



Figure 4: Socket [2]

Modelling

For creating the 3D model of the sphere head and the socket (Fig. 5) we used the results of the 3D measurement of the "etalon" socket and the data appearing in the specifications of the sphere head and the socket. We used the Solid Edge planning software to do the planning as the 3D model formed in this way can be used in the research with the FEM software.

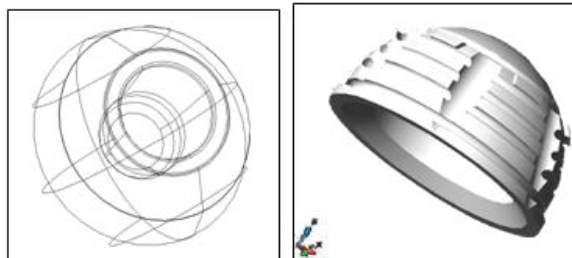


Figure 5: Head and socket models

Finite elements method analysis

The FEM analysis is executed with the help of the so called ANSYS FEM software [3].

Introduction of the ANSYS FEM analysis software

The ANSYS Workbench is the framework upon which the industry's broadest suite of advanced engineering simulation technology is built. An innovative project schematic view ties together the entire simulation process, guiding the user through even complex multiphysics analyses with drag-and-drop simplicity. With bidirectional CAD connectivity, an automated project update mechanism, pervasive parameter management and integrated optimization tools, the ANSYS Workbench platform delivers unprecedented productivity that truly enables Simulation Driven Product Development [4].

The tests were made in two different ways:

1. First we put the sphere head and the socket in each other and some force was applied towards the centre (Fig. 6). After the test we analysed the findings separately for the sphere head and the socket (Figs 7 and 8).

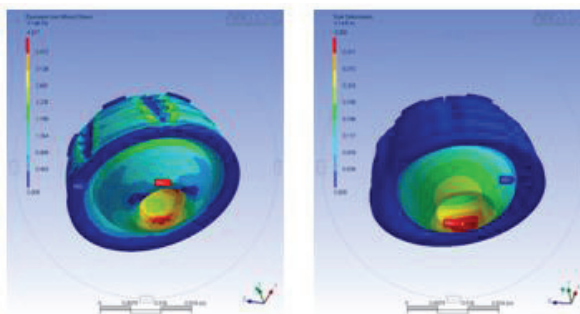


Figure 6: The analysis of tension and deformation, in a complex state

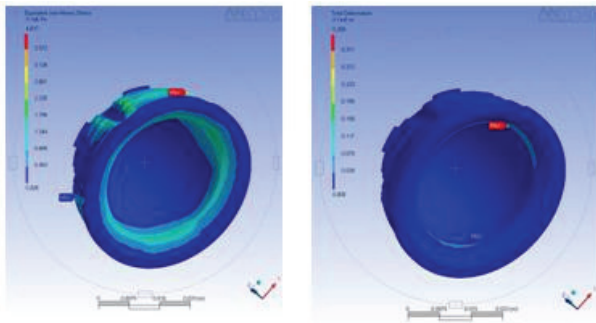


Figure 7: The analysis of tension and deformation on the head

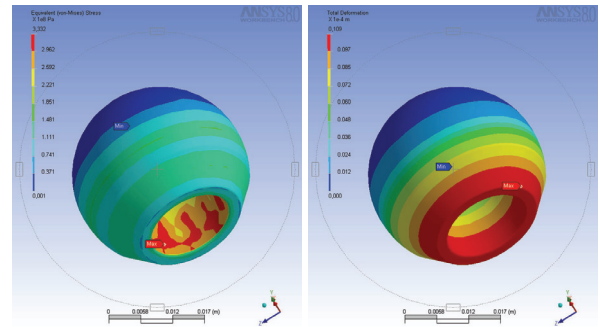


Figure 10: Tension and deformation in the case of the sphere head

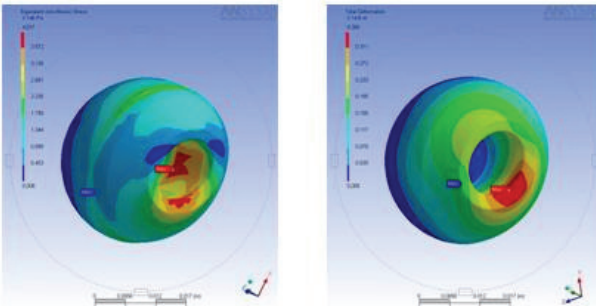


Figure 8: The analysis of tension and deformation on the socket

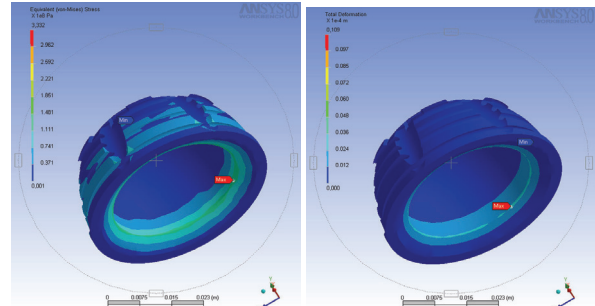


Figure 11: Tension and deformation in the case of the socket

2. In the second stage we applied the torque to the system and we examined tension and values of deformation (Figs 9, 10 and 11).

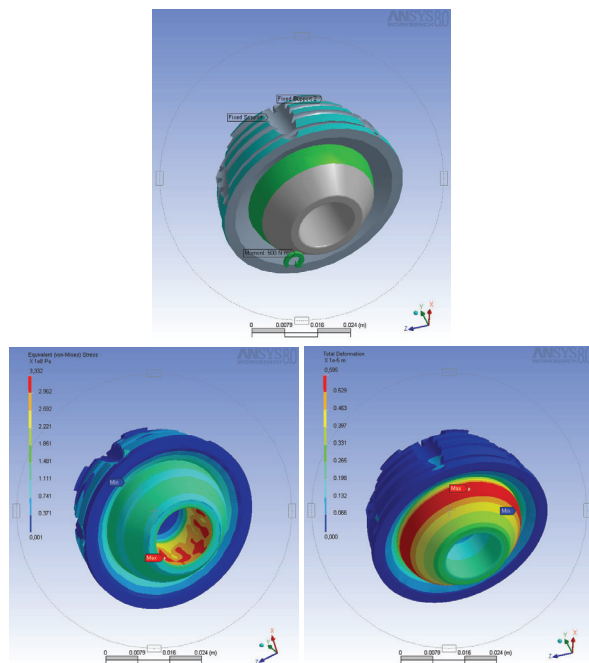


Figure 9: Position of torque and analysis of tension and deformation

Summary

We created the 3D models of the sphere head and the socket with the help of Solid Edge planning software. Using these models we analysed the values of tension and deformation of the sphere head and the socket with the help of the ANSYS FEM software. First, we examined the head and the socket together in each case, and then they were analysed separately. The received results live up to our expectations, that is why later we would like to consider the case when not only one force and one torque is applied but the combination of these (reflecting the reality).

REFERENCES

1. <http://www.metrimed.hu> (2008)
2. <http://www.protetim.hu> (2007)
3. F. SZABÓ, Z. BIHARI, F. SARKA: Products, structures, machinery parts finite element modeling and optimization, Professional Engineering notes, Miskolc 2006
4. <http://www.econengineering.com> (2010)
5. G. HORÁNYI: Life prosthetics (Talking with Prof. Imre Bertóti, in Hungarian). Élet és tudomány, 57(11), (2002), 329–331