Computers in Biology and Medicine 54 (2014) 53–60

Contents lists available at ScienceDirect

Computers in Biology and Medicine

journal homepage: www.elsevier.com/locate/cbm

Design, analysis and verification of a knee joint oncological prosthesis finite element model

Lukáš Zach [a], Lenka Kunčická [b,c,*], Pavel Růžička [a], Radim Kocich [b,c]

[a] Department of Mechanics, Biomechanics and Mechatronics, Faculty of Mechanical Engineering, Czech Technical University in Prague, Technická 4, 166 07 Praha 6, Czech Republic
[b] Department of Materials Forming, Faculty of Metallurgy and Materials Engineering, VŠB-TU Ostrava, 17. listopadu 15, Ostrava-Poruba 70833, Czech Republic
[c] Regional Materials Science and Technology Centre, VŠB-TU Ostrava, 17. listopadu 15, Ostrava-Poruba 70833, Czech Republic

ARTICLE INFO

Article history:
Received 12 March 2014
Accepted 17 August 2014

Keywords:
Endoprosthesis
Oncological implant
Knee joint
Finite element model
Finite element method
Knee-bending

ABSTRACT

Background: The aim of this paper was to design a finite element model for a hinged PROSPON oncological knee endoprosthesis and to verify the model by comparison with ankle flexion angle using knee-bending experimental data obtained previously.
Method: Visible Human Project CT scans were used to create a general lower extremity bones model and to compose a 3D CAD knee joint model to which muscles and ligaments were added. Into the assembly the designed finite element PROSPON prosthesis model was integrated and an analysis focused on the PEEK-OPTIMA® hinge pin bushing stress state was carried out. To confirm the stress state analysis results, contact pressure was investigated. The analysis was performed in the knee-bending position within 15.4–69.4° hip joint flexion range.
Results: The results showed that the maximum stress achieved during the analysis (46.6 MPa) did not exceed the yield strength of the material (90 MPa); the condition of plastic stability was therefore met. The stress state analysis results were confirmed by the distribution of contact pressure during knee-bending.
Conclusion: The applicability of our designed finite element model for the real implant behaviour prediction was proven on the basis of good correlation of the analytical and experimental ankle flexion angle data.

1. Introduction

The knee joint (lat. *articulatio genus*) is a synovial complex joint and is one of the most stressed in the entire human body. The joint consists of three main bones – the thigh bone (*femur*), shin bone (*tibia*) and knee pan (*patella*). Between the femur and tibia bone surfaces can be found the inner and outer meniscuses (*meniscus medialis et lateralis*). The joint further consists of muscles (*musculi*), ligaments (*ligamenta*) and bursas (*bursae*) [1]. Such a complex joint can become severely damaged as a result of illness, disease, injury, accident, etc. In cases of extreme damage, or following degenerative processes, the application of an oncological implant may be necessary. In some cases, unicompartmental knee arthroplasty is satisfactory (e.g. to reform osteoarthritis knee deformation [2]) but the application of a complete endoprosthesis is necessary in most cases [3].

Before implanting a prosthesis into a human body, its behaviour and properties need to be tested to prevent supplemental surgery. Various mechanical tests of oncological implants that may be time and money consuming can be simulated using the finite element method (FEM). FEM simulations have already been a standard development tool in mechanical or materials engineering. In biomechanics they have been used to carry out various analyses of soft tissues, such as simulations of mechanical behaviour of arteries [4] or intervertebral discs [5], or analyses of joints, implants and bones, which are, in orthopaedy, often focused on bone tissue behaviour after an arthroplasty [6–8]. Human joints analyses are mostly carried out to predict mechanical behaviour of a joint [9–11], to investigate stress state in a joint [12], or to find out a reason for pain in the joint [13]. The simulations involving joint implants are often focused on stress analyses between implant parts [14–16] or between the implant and a bone [17]. Implant loading can be analysed from several points of view, such as calculating the entire endoprosthesis

* Corresponding author at: Department of Materials Forming, Faculty of Metallurgy and Materials Engineering, VŠB-TU Ostrava, 17. listopadu 15, Ostrava-Poruba 70833, Czech Republic. Tel.: +420 597 324 433.
E-mail address: lenka.kuncicka@vsb.cz (L. Kunčická).

http://dx.doi.org/10.1016/j.compbiomed.2014.08.021
0010-4825/

strength, locating sites of increased stress concentrations that can lead to implant damage and observing implant behaviour during static and dynamic loading. Moreover, FEM is an important instrument for the computation of contact pressure and its distribution [18,19] that has a direct influence on the development of abrasion between particular parts of the endoprosthesis surfaces; wear being influenced by size and shape of contact areas can be investigated as well [20]. Using the FEM, comparison of different endoprostheses designs [21,22] as well as various prostheses materials [23] can be performed. Advantageously, FEM can be used to accelerate the development of joint implants [24] or to predict new endoprostheses designs [25].

To perform a finite element analysis, first of all, a model must be created. Several papers published elsewhere have already been focused on creation of a finite element model either of a joint itself (knee joint [12,26,27], hip joint [28], temporo-mandibular joint [11], etc.) or of an implant (total knee replacement [29]). Construction of bone models is performed usually on the basis of computer tomography (CT) scans [26,28,30]. Using this method the surface shapes of tissues throughout the body can be displayed by superposition of individual scans, in which different kinds of tissues (e.g. spongy or compact bones) can be distinguished according to the different grey tones. For modelling of soft tissues, such as muscles, ligaments, cartilages or meniscuses, magnetic resonance imaging (MRI) process, suitable for visualisation of tissues with high volume of water, is recommended [26,31].

The aim of this paper was to design a finite element model for a hinged PROSPON oncological knee endoprosthesis. This Czech company, one of the manufacturers of oncological implants, was established in 1992 and focuses on the production of oncological implants and instruments for traumatology and orthopaedic surgery (www.prospon.cz). Its implants are widespread all over the central Europe, and also in France, Turkey, Greece or Russia. The knee endoprostheses are non-symmetrical and case-hardened and are basically made of the Ti–6Al–4V alloy, which is among the basic materials used in the manufacture of implants [32,33] together with titanium–nickel (shape memory) alloys [34–37], the Co–Cr–Mo alloy [2,23], or ceramics [23]. The prostheses condyles are coated with a DLC (diamond-like carbon) layer. Bone replacements are made from UHMWPE, ultrahigh molecular weight polyethylene, while the PEEK-OPTIMA® material is used for the sliding hinge pin bushings. Although the PEEK-OPTIMA® is commonly used by various companies for spine implants (SIGNUS Medical; Calvary Spine Products, etc.), it is not so widespread among joint prostheses. The designed finite element prosthesis model was implemented into a lower extremity model created on the basis of Visible Human Project CT scans to carry out a finite element simulation focused on the hinge pin bushing stress state analysis; the pin bushing had been considered to be the most loaded part of the implant. A knee-bending position was chosen due to the presupposition of increased knee endoprosthesis loading. Verification of the model was done by comparison with previously obtained ankle flexion angle in knee-bending experimental data [3].

2. Material and methods

2.1. Oncological implant

The PROSPON oncological implants are made to order for an individual patient; the used oncological implant consisted of the following parts: femoral stem, femoral component, rotating hinge post, tibial plateau, tibial base plate, tibial stem, medial/lateral hinge pin bushing, hinge pin, and hinge pin plug. The implant is schematically depicted in Fig. 1.

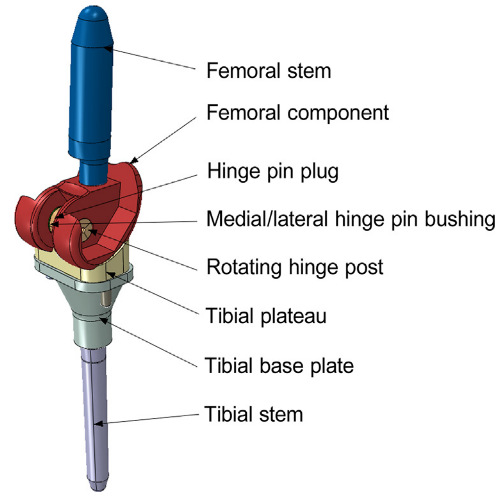

Fig. 1. The complete geometrical model of the PROSPON oncological implant.

2.2. 3D model

For the 3D reconstruction of lower extremity bones in this analysis, scans acquired within the Visible Human Project (National Library of Medicine) using the CT method were used (http://www.nlm.nih.gov/research/visible/visible_human.html). In this research project, a man 186 cm in height and 90 kg in weight had been scanned and the scans had been saved at 1 mm intervals with a resolution of 512 × 512 pixels. After acquisition, CT scans had to be adjusted and segmented. During segmentation, individual types of tissues were assigned to individual regions of the scans according to the grey tones; final manual correction of the boundary lines was necessary. The 3D reconstruction process, during which a 3D model from the individual 2D scans of the tissue boundary lines was created, followed. A 3D model was constructed semi-automatically and its boundary (surface) consisted of a mesh of triangles. Such a model was then ready to be exported in the VRML (Virtual Reality Mark-Up Language) format. The VRML file assembled from the scans had to be further adjusted and modifications related to implant integration had to be done. During this adjustment process the mesh of triangles was transformed into a set of surfaces from which a 3D lower extremity CAD model, which was used for the analysis of the PROSPON knee endoprosthesis loading, was created. For the creation of a CAD model from the VRML file, UNIGRAPHICS NX software with the GRIP program kit upgrade was used. According to the operating procedures of the manufacturer, setting of the knee endoprosthesis on the relevant bones and necessary final verification of the implant by the manufacturer and an experienced physician were carried out.

To carry out a realistic 3D FEM simulation of knee endoprosthesis loading the main lower extremity muscles were added to the assembly. For this analysis, works published by White et al. [38] were used as primary data sources. Pelvis geometry was added to the model in order to implement the coordinates of the muscle tendons. For this purpose the pelvis geometry model VRML file, created by Viceconti at the Biomechanics European Lab [39] also using the Visible Human Project CT scans, was incorporated. According to White, 25 main muscles were implemented: *Mm. iliacus, psoas; M. Pectineus; M. adductor longus; M. adductor brevis; M. gluteus maximus; M. gluteus medius; M. gluteus medius posterior; M. gluteus minimus anterior; M. gluteus minimus posterior; Biceps femoris longus; M. semitendinosus; M. semimembranosus; M. Sartorius; M. gracilis; Tensor fasciace latae; Rectus femoris; Biceps femoris brevis; Vastus lateralis; Vastus intermedialis; Vastus medialis; M. gastrocnemius medialis; M. gastrocnemius lateralis; M. soleus; M. tibialis anterior;* and *M. tibialis posterior* [3]. The coordinates