Available online at www.sciencedirect.com
ScienceDirect
Journal of Bionic Engineering 10 (2013) 350–358

A Femur-Implant Model for the Prediction of Bone Remodeling Behavior Induced by Cementless Stem

He Gong[1,2], Lingyan Kong[2], Rui Zhang[2], Juan Fang[2], Meisheng Zhao[3]

1. *State Key Laboratory of Automotive Simulation and Control*, *Jilin University*, *Changchun* 130025, *P. R. China*
2. *Department of Engineering Mechanics*, *Jilin University*, *Changchun* 130025, *P. R. China*
3. *Department of Cataract*, *Eye Hospital of the Second Clinical Hospital of Jilin University*, *Jilin University*, *Changchun* 130051, *P. R. China*

Abstract

Bone remodeling simulation is an effective tool for the prediction of long-term effect of implant on the bone tissue, as well as the selection of an appropriate implant in terms of architecture and material. In this paper, a finite element model of proximal femur was developed to simulate the structures of internal trabecular and cortical bones by incorporating quantitative bone functional adaptation theory with finite element analysis. Cementless stems made of titanium, two types of Functionally Graded Material (FGM) and flexible 'iso-elastic' material as comparison were implanted in the structure of proximal femur respectively to simulate the bone remodeling behaviors of host bone. The distributions of bone density, von Mises stress, and interface shear stress were obtained. All the prosthetic stems had effects on the bone remodeling behaviors of proximal femur, but the degrees of stress shielding were different. The amount of bone loss caused by titanium implant was in agreement with the clinical observation. The FGM stems caused less bone loss than that of the titanium stem, in which FGM I stem (titanium richer at the top to more HAP/Col towards the bottom) could relieve stress shielding effectively, and the interface shear stresses were more evenly distributed in the model with FGM I stem in comparison with those in the models with FGM II (titanium and bioglass) and titanium stems. The numerical simulations in the present study provided theoretical basis for FGM as an appropriate material of femoral implant from a biomechanical point of view. The next steps are to fabricate FGM stem and to conduct animal experiments to investigate the effects of FGM stem on the remodeling behaviors using animal model.

Keywords: functionally graded material, bone remodeling, stress shielding, proximal femur, implant

doi: 10.1016/S1672-6529(13)60230-9

1 Introduction

Bone's mass and architecture are adapted to the mechanical environment, as well as other biophysical stimuli, which is known as functional adaptation[1,2]. Bones change their mass and architecture following the change of external mechanical environment to make them still adapt to the new mechanical environment. Nowadays, with the development of heavy computational technology, it is possible to quantitatively simulate the bone adaptation process in order to further predict and explain the formation and maintenance of bone architecture under different mechanical environments by incorporating quantitative bone remodeling algorithm with finite element analysis[2–4].

The incidence of fractures has increased considerably in recent years with the rapid increase in aging population. It was estimated that one in two women and one in five men over the age 50 would suffer a fracture in their remaining life time[5]. Hip fracture is one of the most serious types of fractures. The treatment of hip fracture requires a fixation device to resume the weight-bearing ability. When an implant (that is usually made of titanium alloy) is introduced, it will bear a portion of the load originally carried by bone tissue, thus reducing the mechanical load in some regions of the host bone. This phenomenon is commonly called stress shielding. The decrease in the mechanical load will trigger bone remodeling process to change its mass and architecture. With quantitative bone remodeling simulation method the bone remodeling process caused by femoral implant can be simulated[6]. Furthermore, the

Corresponding author: Meisheng Zhao
E-mail: zhaoms1961@sina.com

effects of materials of femoral stem on the bone remodeling behaviors can be investigated[7].

However, in our earlier study, the finite element models of the cementless hip prosthesis and femur were simple, and the loading conditions were also simplified[7]. As a result, it is difficult to describe the adaptive behaviors of femur before and after hip replacement consecutively and adequately. Previously, we simulated the external shape and internal structure of a two-dimensional proximal femur[8]. But the height of that model was not large enough to accommodate a stem. Accordingly, the purposes of this study were:

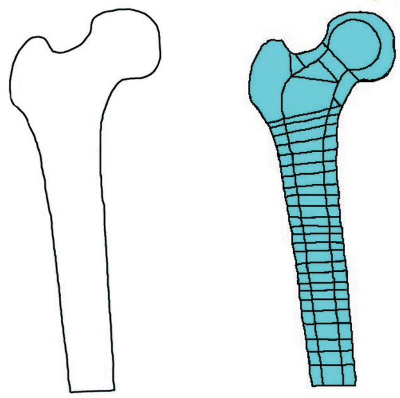
(1) to develop a finite element model of proximal femur with the loading condition taking into account different phases in a gait cycle. It can be used to simulate the internal structures of trabecular bone and cortical bone;

(2) to compare the stems made of different materials, *i.e.* titanium, Functionally Graded Material (FGM) with two kinds of basic constituent materials, and flexible 'iso-elastic' material on the bone remodeling behaviors of host bone.

2 Materials and methods

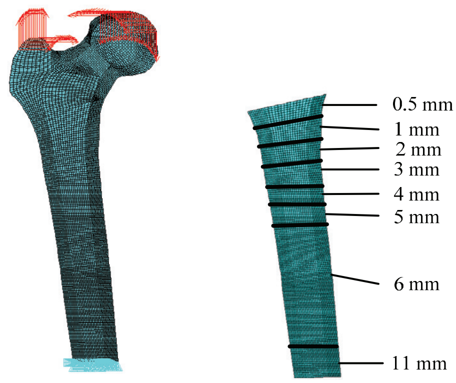
2.1 Finite element model of the two-dimensional proximal femur

Contour of the coronal plane of proximal femur was obtained from the reconstruction of a series of CT scanning images, as shown in Fig. 1a. Then it was filled to PLANE entity using Solidworks (Dassault systems Solidworks Corp.) and imported to ANSYS (ANSYS Inc). Mapped meshing was used to mesh the model with four-node bilinear elements. Since the shape of femur model was irregular, the model was divided into several parts with regular shapes. Since the remodeling behavior of the host bone after hip replacement would be simulated followingly, the shape of hip prosthesis was included in the divisions, which can be clearly seen in Fig. 1b. The mesh of the femur model was shown in Fig. 2a with 8716 plane stress elements and 8972 nodes. Since the model in this paper was two-dimensional, a side plate was used to simulate the three-dimensional connectivity[9]. The side plate was connected to the front plate by the medial and lateral cortical bones, and the thickness of side plate changed gradually, *i.e.* the thickness was 0.5 mm, 1 mm, 2 mm, 3 mm, 4 mm, 5 mm, 6 mm and 11 mm from top to bottom as shown in Fig. 2b. The mesh of the side plate was the same with that part of the front plate. Including the side plate, the model totally comprised 14451 plane stress elements and 14588 nodes. Bone remodeling algorithm was only applied to the front plate. The material property of the side plate was unchanged during simulation with Young's modulus of 14217 MPa and Poission's ratio of 0.3, which were the same as cortical bone.



(a) (b)

Fig. 1 Two-dimensional model of the proximal femur. (a) Contour of the coronal plane of proximal femur; (b) divisions made for the mapped meshing.



(a) (b)

Fig. 2 Finite element model of the proximal femur. (a) Mesh of the femur model with loading and boundary conditions; (b) finite element model of the side plate.

Physiological loading was applied with three loading cases representing three phases of the gait cycle, *i.e.* the loading of the femur at 10% (heel strike), 30% (mid stance) and 45% (push off) of the gait cycle[10]. The loading was shown in Table 1. Femoral head was exerted hip joint contact force, and great trochanter was exerted muscle forces. Those forces were applied in the parabola form. In the present study, the muscle forces caused by