Medical Engineering & Physics 34 (2012) 914–919

Contents lists available at SciVerse ScienceDirect

Medical Engineering & Physics

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

Orientation of orthotropic material properties in a femur FE model: A method based on the principal stresses directions

T. San Antonio [a,b,*], M. Ciaccia [a,b], C. Müller-Karger [a], E. Casanova [a]

[a] Grupo de Biomecánica, Universidad Simón Bolívar, Caracas, Venezuela
[b] Centro de Investigaciones en Mecánica (CIMEC), Facultad de Ingeniería, Universidad de Carabobo, Valencia, Venezuela

ARTICLE INFO

Article history:
Received 28 February 2011
Received in revised form 20 October 2011
Accepted 22 October 2011

Keywords:
Orthotropic axes orientation
Principal stress
Wolff's law
Proximal femur
Subject-specific bone FE model

ABSTRACT

Most work done on bone simulation has modeled the tissue as inhomogeneous and isotropic even though it is a recognized anisotropic material. Some recent investigations have included orthotropic behavior in bone finite elements (FE) models; however the problem regarding the orientation of these properties along the irregular bone anatomy remains. In this work, a procedure to orientate orthotropic properties in a proximal femur FE model using the directions of the principal stresses produced by a physiological load scheme was developed. Two heterogeneous material models, one isotropic and one orthotropic, were employed to test their influence on the mechanical behavior of the bone model. In the developed orthotropic material, the mechanical properties are aligned with the highest principal stress produced from the successive application of a multi load scenario corresponding to 10%, 30% and 45% of the gait cycle. A solid match between anatomical structures in the proximal femur and the corresponding directions of the main principal stress of the elements of the model suggests that the developed methodology works accurately. The differences found in the stress distributions were small (maximum 7.6%); nevertheless the changes in the strain distributions were important (maximum 27%) and located in areas of clinical relevance.

1. Introduction

The development of subject-specific finite element (FE) models using computed tomography (CT) data is a powerful tool to non-invasively investigate clinical applications such as fracture risk, prosthesis design and bone remodeling [1–3]. Most work done in the simulation of bone tissue have assumed the material to behave as inhomogeneous and isotropic due to the simplicity of its implementation; some of these models have been reported to properly reproduce the strains and displacements on the surface when compared with experimental results [4,5]. However, bone tissue is widely recognized as an anisotropic material due to its structural patterns and mechanical behavior. From the literature it is well known that the cortical bone exhibits at least transverse isotropy in terms of its elastic properties [6] while the trabecular bone is effectively anisotropic [7,8].

In recent years there has been some investigation including the orthotropic behavior of the bone tissue in the FE models; however the central problem regarding the orientation of these properties along the irregular anatomy of bones remains. Even though some researchers have used models with non oriented orthotropic properties [9,10], most studies do orientate the properties using few anatomical directions corresponding to the bone shape [10–13], by means of the variation of the CT Hounsfield units values (density) field [14,15] or using invasive methods that involve slicing or crushing the bones to find the directions of the axes of orthotropy [16,17]. However, bone remodeling is the most widely used method to obtain, in an iterative process, oriented orthotropic properties along with density distributions based in a specific load case [18–22]; in contrast, with the methodology developed in this work the density distribution is readily obtained from the CT images, allowing for subject-specific bone models, and the orthotropic properties are oriented with a non iterative procedure that requires lower computational resources.

The orientation of the actual orthotropic material axes does not necessarily correspond to a reference frame or a prefixed trajectory throughout the bone geometry, being difficult to establish when the bone shape is irregular. However, "Wolff's law" proposed that trabeculae patterns coincide with the directions of the principal stresses. This is justified from an analytical criterion, since effective material properties such as stiffness and strength are higher in these directions, and it is intuitively clear that the

* Corresponding author at: Centro de Investigaciones en Mecánica (CIMEC), Facultad de Ingeniería, Universidad de Carabobo, Valencia, Venezuela.
Tel.: +58 2418723656.
E-mail address: tsantoni@uc.edu.ve (T. San Antonio).

1350-4533/$ – see front matter © 2011 IPEM. Published by Elsevier Ltd. All rights reserved.
doi:10.1016/j.medengphy.2011.10.008

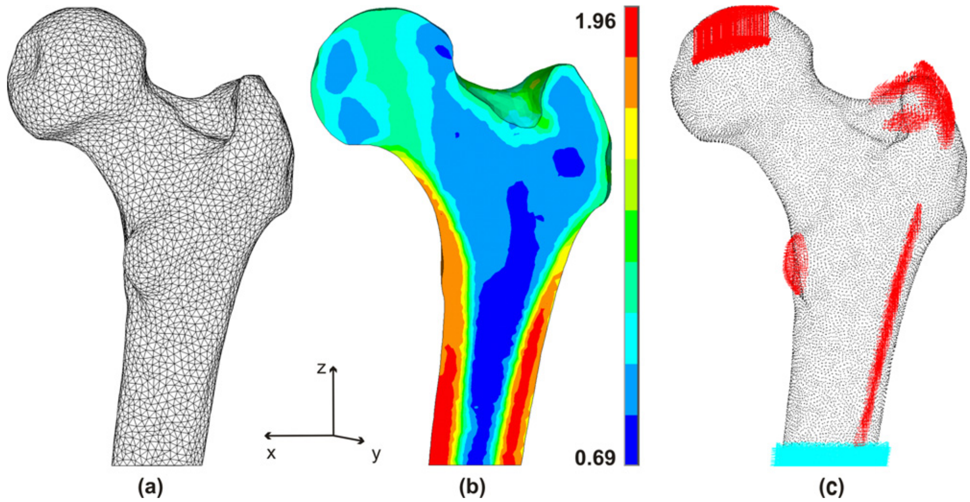

Fig. 1. Proximal femur model, (a) mesh with 99,614 10 nodes tetrahedral elements, (b) slice showing the internal density distribution (g/cm^3), (c) external nodes showing the application of loads (red) and constrains (cyan). (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

strongest and stiffest material axis should be in the direction of the largest stress. Early interpretations of "Wolff's law" have assumed that the principal stress correspond to a single average load applied to the bone; however subsequent works have shown that a single load cannot predict the trabecular structure correctly and a multiple load set must be used; it has also been suggested that trabecular pattern is defined by extreme loading directions [19].

The first objective of this study was to develop a procedure to orientate orthotropic properties on each element of the FE model using the direction of the maximum principal stress due to a multiple physiological load scheme. The second objective was to assess the influence of the material axes orientation in the mechanical behavior of a subject-specific proximal femur FE model.

2. Materials and methods

2.1. Finite element model

The CT images used to build the FE model of the proximal femur are from a 38 years old male, 1.77 m of stature and 95 kg of weight, free of bone related diseases. The scans of 256×256 pixels were taken at 1 mm intervals, and the pixel size is 0.48 mm $\times$ 0.48 mm. The heterogeneous model is 154 mm long and was constructed following three steps: (1) three-dimensional reconstruction of the proximal right femur using CT images; (2) generation of the FE meshes using ANSYS and its 10 nodes tetrahedral solid element SOLID92. In order to check for convergence, eight meshes were built with the element edge size ranging from 7 to 2.5 mm. The selected mesh, used with all material models, consists of 99614 elements and 143029 nodes, and was found to have sufficiently converged giving a strain energy gradient of 8.6×10^{-6}% per unit increase in degree of freedom [23] (see Fig. 1a); (3) the Hounsfield unit (HU) value was determined at four Gauss points for every element in the FE model by relating its coordinate position to the CT scan data, and then translated to bone apparent density by means of the tomograph calibration curve. It has been verified that the convergence of the von Mises stress improves when the density of each 10 nodes tetrahedral element is calculated as a weighted average of this property in at least four Gauss points [24]. Apparent density $\bar{\rho}_e$ was then computed with Eq. (1).

$$\bar{\rho}_e = \frac{\int_{V_e} \rho(x, y, z) dV}{V_e} = \frac{\sum_{i=1}^{4} w_i \rho(\zeta_i, \eta_i, \xi_i) \det \mathbf{J}(\zeta_i, \eta_i, \xi_i)}{\sum_{i=1}^{4} w_i \det \mathbf{J}(\zeta_i, \eta_i, \xi_i)} \quad (1)$$

where V_e indicates the volume of the element e, (x, y, z) are the coordinates in the CT reference system, (ζ_i, η_i, ξ_i) are the local coordinates of the Gauss points in the element reference system, w_i is the weight for Gaussian numerical integration, ρ is the apparent density and $\mathbf{J}$ represents the Jacobian of the transformation. Fig. 1b shows the internal density distribution of the proximal femur model.

The computed mesh and density distribution were used to perform two simulations which had identical boundary conditions but different material properties. The simulation with the isotropic material was set up using a standard approach, which consists in assigning material properties, imposing boundary conditions and then solving the numerical model. The orthotropic material model followed a more elaborated procedure, requiring the assignation of orthotropic material properties in an arbitrary direction, and then solving for three different sets of loads in order to define the final directions of the axes of orthotropy. With these directions defined and applied to each element, a final solution of the model was obtained for the set of boundary conditions of interest.

2.2. Boundary conditions

A common simplification done to the boundary conditions in the proximal femur FE models is to represent only the hip joint contact force and adductor muscle force, acting on the femur head and the greater trochanter respectively [9,13,17,19].

In this study a complex loading configuration, comprising seven muscle forces and the hip joint contact force (see Fig. 1c), was used in order to reproduce the physiological conditions of the proximal femur [25]. The components of the forces shown in Table 1 are expressed in the global coordinate system of the FE model used in this study, with the Z direction parallel to the longitudinal femur axis and the XZ plane passing through the femur head and the great trochanter.