A Parametric Analysis Study on the Number of Materials Required for a Convergence of Finite Element Results for a Tibial Bone Model

K Biegler*, J Schmidt, H Ploeg
Department of Mechanical Engineering, University of Wisconsin-Madison, Madison, WI
* 1550 University Avenue, Madison, WI 53706,Tel: 608-263-6692, Fax: 608-265-2316, kkbiegler@wisc.edu

INTRODUCTION

Early Finite Element (FE) models of a proximal tibial bone implanted with a tibial component of a Total Knee Replacement (TKR) employed two-dimensional and axisymmetric models. These models provided useful information for early implant development, but three-dimensional models are required to capture a more accurate structural response. The first three-dimensional FE models represented cortical and cancellous bone as homogenous materials with isotropic and linear elastic material properties. The next development in generating FE models of bone introduced non-homogenous material representation of bone, by modeling discrete sections of different but homogeneous materials within the geometric representations of the bone [2].

Computed Tomography (CT) scan data has been studied extensively as a method to determine the apparent density of bone. Continued research is being performed to draw more accurate relationships between the apparent density of bone and the mechanical properties. Using defined equations published in research journals, material properties can be assigned to an FE bone model based on the CT number distribution of the bone. Determining the minimum required number of discrete material properties to accurately represent the bone is an initial step in developing a usable and accurate FE model of the bone, and therefore, the purpose of this study.

METHODS

FE bone models were created in ABAQUS 6.4 (Abaqus Inc., Providence, RI) of the left tibia of the Visible Human male data set (United States National Library of Medicine, Bethesda, MD). Quadratic tetrahedral elements with a global edge length of 3 mm were used to create the FE mesh. A protocol to create FE models from CT data was used to determine both the geometric representation of the bone and to assign material regions based on the CT number [1].

A literature review was performed to determine the appropriate equations relating CT number to apparent density and apparent density to Young's modulus. Various relationships have been determined experimentally by several research groups (see Figure 1). These relations are used to assign the location and value of the discrete material properties to the FE mesh, based on the CT numbers.

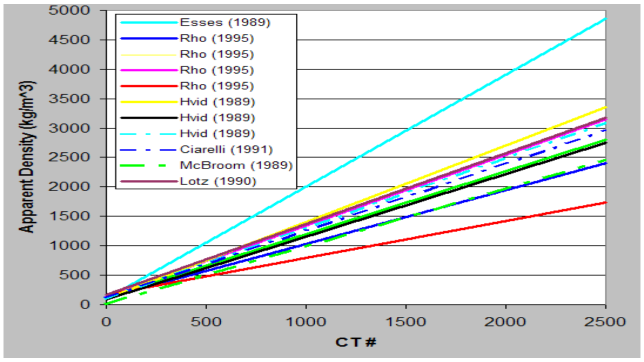

Figure 1 *Correlations found relating the CT number from the CT data to the apparent density of the tibial bone. Similarly, results were found for the correlation between apparent density and axial Young's modulus.*

Selected from the available correlations found in the literature review, the correlates for both the apparent density and axial Young's modulus were chosen based on the smallest square of the correlation coefficient value. The next step was to choose the discrete number of material properties to be assigned to the bone model. A preliminary range was selected based on the minimum required and a reasonable maximum number of material regions based on FE software capabilities. 2, 5, 10, and 20 discrete material properties for the FE model of the tibial bone where chosen for the initial experimental setup.

Mimics 9.0 (Materilise, Ann Arbor, MI) is used to apply the material property relationships, create the discrete number of material properties, and apply them to the FE model created from ABAQUS based on the CT number. Using the average CT value that corresponds to the volume of a single FE element a material will be assigned. This allows for material properties to be assigned on an element by element base. Setting the number of materials in the model only limits the number of available materials that can be assigned to the element. This allows for a distribution of materials that better represent the true material distribution found in bone.

For all material cases a force of 2000 N was applied evenly between the medial and lateral compartments. This load case provides a comparison to other research to help validate the method of material assignment and the results from the FE analysis.

ANALYSIS

Preliminary results demonstrated a change in the response of the implant stresses, both location and magnitude, due to the number of material regions associated to the bone model. Micromotion at the implant-to-bone interface was also affected by the number of material regions (see Figure 2).

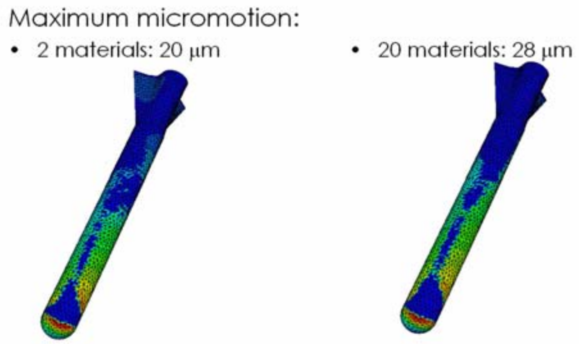

Figure 2 *Preliminary results showing the micromotion between the stem and bone interface. Changes with the maximum micromotion value can be seen as the number of material regions varies.*

Further analysis was performed that included an in depth look at the material placement within the model and the effects they have on stress concentrations and accuracy. Based on the more detailed study of the FE analysis results further cases were run with a more extensive range of the number of material properties assigned. Varying degrees of heterogeneity have been represented in FE models but the accuracy of their solutions may be improved by the representation of anisotropy [2]. To explore the validity of this statement, the final step of this study was to use published relationships for the material properties of tibial bone in both axial and transverse directions to create a FE model of a bone with heterogeneous transverse isotropic material properties. The resulting FE bone model is used to explore the coupled effects of the number of material properties and the material model on the FE results.

REFERENCES

1. Schmidt J, Henderson A, Ploeg H, Deluzio K, Dunbar M submitted to *14th Annual Symposium on Computational Methods in Orthopaedic Biomechanics*. 2005. Chicago, IL.
2. Au, A. G.,Liggins, A. B.,Raso, V. J.,Amirfazli, A.,2005. *A parametric analysis of fixation post shape in tibial knee prostheses.* Medical Engineering and Physics 27. 123-134