



## Technical note

# Comparisons of node-based and element-based approaches of assigning bone material properties onto subject-specific finite element models

G. Chen<sup>a,\*</sup>, F.Y. Wu<sup>a</sup>, Z.C. Liu<sup>a</sup>, K. Yang<sup>a</sup>, F. Cui<sup>b</sup><sup>a</sup> School of Civil and Transportation Engineering, Guangdong University of Technology, Guangzhou 510006, China<sup>b</sup> Institute of High Performance Computing, A\*STAR, Singapore

## ARTICLE INFO

## Article history:

Received 21 January 2015

Revised 8 April 2015

Accepted 2 May 2015

## Keywords:

Bone  
 Computed tomography  
 Finite element models  
 Material assignment  
 Node-based approaches  
 Element-based approaches

## ABSTRACT

Subject-specific finite element (FE) models can be generated from computed tomography (CT) datasets of a bone. A key step is assigning material properties automatically onto finite element models, which remains a great challenge. This paper proposes a node-based assignment approach and also compares it with the element-based approach in the literature. Both approaches were implemented using ABAQUS. The assignment procedure is divided into two steps: generating the data file of the image intensity of a bone in a MATLAB program and reading the data file into ABAQUS via user subroutines. The node-based approach assigns the material properties to each node of the finite element mesh, while the element-based approach assigns the material properties directly to each integration point of an element. Both approaches are independent from the type of elements. A number of FE meshes are tested and both give accurate solutions; comparatively the node-based approach involves less programming effort. The node-based approach is also independent from the type of analyses; it has been tested on the nonlinear analysis of a Sawbone femur. The node-based approach substantially improves the level of automation of the assignment procedure of bone material properties. It is the simplest and most powerful approach that is applicable to many types of analyses and elements.

© 2015 IPEM. Published by Elsevier Ltd. All rights reserved.

## 1. Introduction

Subject-specific finite element (FE) models have been commonly employed to study the mechanical behaviours of bone structures. Both model geometries and material properties of subject-specific FE models can be automatically generated from computed tomography (CT) datasets of a bone. Subject-specific FE models can be created by two strategies: (1) voxel-based and (2) geometry-based [1,2]. For voxel-based models, FE datasets, including bone material properties, are directly generated from CT images using a commercial code, e.g., Simpleware (Simpleware Ltd., Exeter, UK). The apparent disadvantages of this strategy include low accuracies in FE model geometries and bone material properties [2]. The geometry-based strategy is more commonly used in bone simulations [1], and the standard procedures for this strategy were reported, for example, in Taddei et al. [3] and Kluess et al. [4]. Generation of FE model geometries can be realised using a couple of commercial software packages unitedly, as reported in Messmer et al. [5]; however, assigning material properties onto FE models usually needs an in-house program code (e.g., BONE-MAT) to connect the CT dataset and FE datasets [6,7]. The automation

of assigning material properties is influenced by the implementation of assignment approaches [8] and great efforts have been made to improve the level of assignment automation and accuracy of material properties [3,4,6,7,9–12].

Several assignment approaches have been reported for mapping bone material properties from CT images onto finite element models for stress analyses [4,6–9,13,14]. In the early stage, the researchers developed in-house assignment programs [7,14], which usually connect the CT dataset and FE dataset to assign the Young's modulus to each element (i.e., element-based). As each element type has a different data structure in the FE models, hence the element-based assignment approach is dependent on the type of elements used; consequently, the implementation of this approach needs complex programming. Obviously the implementation of the element-based approach is also dependent on the type of analyses, because a different analysis owns a different data structure in the FE models. To make the assignment process independent from the type of elements, Helgason et al. [8] and Kluess et al. [4] assigned the intensity value of CT images as the “temperature” at the FE nodes (i.e., node-based) and made the bone material properties (i.e., Young's modulus) dependent on the “temperature”. This approach is independent from the type of elements but dependent on the type of analyses. The obvious disadvantage of all the above-mentioned approaches is that, assignment of material properties from the CT dataset to a FE model of a bone involves

\* Corresponding author. Tel.: +86 136 6248 3527

E-mail address: [gongfa.chen@gdut.edu.cn](mailto:gongfa.chen@gdut.edu.cn) (G. Chen).

complex programming, as these methods directly modify the FE dataset.

Recently, the assignment procedure has been substantially simplified by Chen et al. [9]. Instead of modifying the dataset of FE models, their assignment approach resorts the special functionality of commercial codes and assigns the inhomogeneous material properties via a user-defined material. Their assignment procedure is divided into two steps: generating the data file of the image intensity of a bone in a MATLAB (MathWorks, Inc., Natick, MA, USA) program and reading the data file into ABAQUS (SIMULIA, Inc., Providence, USA) via user subroutines. The material properties are directly assigned to each integration point of an element (element-based). This assignment approach is independent from the type of elements [9]; however, it is dependent on the type of analyses. For a normal linear elastic analysis, the approach involves very little programming; nevertheless, for complex analyses, e.g., plastic analyses, the users may have to develop complicated subroutines to realise their analyses. To avoid complicated programming, an assignment approach, which is independent from the type of elements and the type of analyses, is desired.

This paper proposes a node-based approach to assign bone material properties, in which material properties are defined on the FE nodes by using a field variable and the field variable is calculated from the Hounsfield units (HUs) of bone CT images. It also incorporates the idea of splitting the assignment procedure into two steps [9], i.e., the image intensity of a bone is generated in a MATLAB program and saved in a data file which is read into ABAQUS via user subroutines. The node-based approach will be tested on different mesh types and different analyses (linear and nonlinear). The assignment accuracy of this approach will be verified by comparisons with the element-based approach proposed by Chen et al. [9].

## 2. Methods

A Sawbone femur model (Pacific Research Laboratories, Inc., Washington, United States) was scanned using a Brilliance 64 CT scanner (Philips, Best, the Netherlands), operating at 120 KVP with a  $0.23 \times 0.23$  mm in-plane pixel size and with a slice thickness of 0.67 mm. The images were saved in 526 DICOM files.

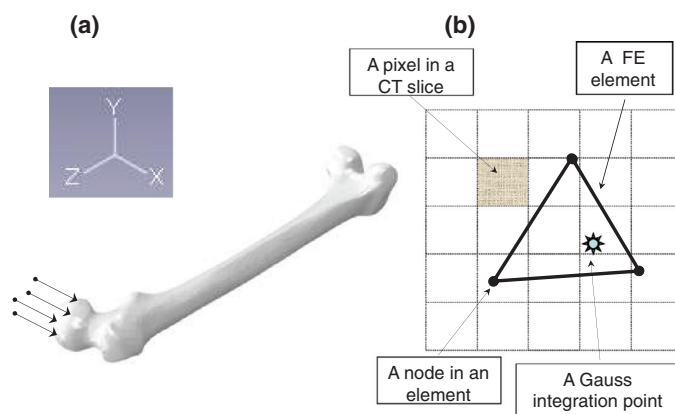
Both the geometry and material properties of a FE model were derived from the CT dataset of the Sawbone femur.

### 2.1. Acquisition of bone density from CT images

All CT images of the Sawbone femur were first processed by an in-house MATLAB program [9] to obtain the intensity of the CT images and the values were saved into a data file. In the MATLAB program, three tasks were performed: (1) 2D segmentation of bone for each slice; (2) correction of partial volume effect (PVE) on the boundary pixels; (3) output of the pre-processed images and their HUs. To extract an accurate FE model geometry from CT images, a multiple-threshold segmentation method was used. The FE model geometry generated in this segmentation method is more accurate than that created by single-threshold segmentation. The multiple-threshold segmentation techniques and correction of PVE used in this paper had been reported in Chen et al. [9]. The pre-processed CT images were used for creation of Sawbone geometry in Amira (Mercury Computer Systems, Inc., Chelmsford, USA) and the HUs of CT images were saved into a TEXT file, which will be read by ABAQUS.

### 2.2. Creation of FE model geometry from CT images

A standard procedure to create virtual 3D models of the bones, using commercial software packages, has been reported in Messmer et al. [5]. First, the visualisation software Amira was used to extract the external contour of the Sawbone femur from the pre-processed



**Fig. 1.** Assigning material properties from CT dataset to a FE model. (a) The FE geometry imported into ABAQUS. The arrow indicates the loading direction. (b) Assigning material properties from CT dataset to a Gauss integration point or a node.

CT images; the results were saved in a STL file. This surface model was then converted into a NURBS model using the software Rapidform (INUS Technology, Seoul, Korea). A solid was then created from the NURBS model in Rapidform and saved as an IGES file. This IGES file was imported by ABAQUS for creation of the geometry of the FE model (Fig. 1a).

### 2.3. Finite element model

The femur model was fixed at the distal end and subjected to lateral forces of 400 N at the proximal end along the X direction (Fig. 1a). To avoid apparent stress concentration, the loads were applied at the four nodes which were located at  $(-105.5, 158.4, 508.6)$ ,  $(-96.7, 168.4, 536.1)$ ,  $(-93.2, 140.5, 546.8)$ , and  $(-102.1, 130.1, 519.2)$ . The four-node tetrahedral elements were used for the major part of the bone and the average aspect ratio was 1.6. Both the four-node tetrahedral elements and eight-node hexahedral elements were tested in the middle part of the bone where the comparisons were made. Two different element sizes, rough (2.0 mm) and fine (1.0 mm), were used for stress analysis.

For the nonlinear stress analysis, the yield stress of the cortical part was  $\sigma_y = 198.0$  MPa, the tangential modulus was  $E_p = 0.14E$ . The von Mises yield criterion and isotropic hardening law were used [15]. The loading position and direction were kept as above and the value of the load was doubled.

The material properties were assigned by either the element-based or node-based assignment approaches onto the FE models during analyses, which were explained in the following.

### 2.4. Assignment of material properties to finite element model

ABAQUS read in the text file which saves the HUs via a subroutine "UEXTERNALDB" at the beginning of the analysis to calculate the material properties of the Sawbone femur.

- Element-based assignment approach [9]. A user-defined material was assigned for the bone. For each Gauss integration point, the pixels enclosing this point were searched (Fig. 1b) during the analysis and the average HU of the surrounding  $(3 \times 3 \times 3)$  pixels was calculated. This average HU was assigned to a solution-dependent state variable as the initial value at this integration point through a user-subroutine "SDVINI" in ABAQUS. The values of the solution-dependent state variable at integration points were used to evaluate the Young's modulus in the user-subroutine "UMAT", which calculates the density, stiffness, strain, and stress at each integration point [16].

Download English Version:

<https://daneshyari.com/en/article/875680>

Download Persian Version:

<https://daneshyari.com/article/875680>

[Daneshyari.com](https://daneshyari.com)