Journal of Biomechanics 45 (2012) 2702–2705

Contents lists available at SciVerse ScienceDirect

Journal of Biomechanics

journal homepage: www.elsevier.com/locate/jbiomech
www.JBiomech.com

Short communication

Finite element analysis of a micromechanical model of bone and a new 3D approach to validation

S.P. Evans [a], W.C.H. Parr [b], P.D. Clausen [a], A. Jones [c], S. Wroe [b,*]

[a] School of Engineering, University of Newcastle, Newcastle, NSW 2038, Australia
[b] School of Biological, Earth and Environmental Sciences, University of New South Wales, Sydney, NSW 2052, Australia
[c] Australian Centre for Microscopy and Microanalysis, University of Sydney, Sydney, NSW 2006, Australia

ARTICLE INFO

Article history:
Accepted 2 August 2012

Keywords:
Finite Element Analysis
Validation
Trabecular network

ABSTRACT

Finite Element Analysis (FEA) is now widely used to analyse the mechanical behaviour of bone structures. Ideally, simulations are validated against experimental data. To date, validation of Finite Element Models (FEMs) has been 2 Dimensional (2D) only, being based on comparison with surface-mounted strain gauge readings. In this study we present a novel 3-Dimensional (3D) approach to validation that allows comparison of modelled with experimental results between any two points in 3D space throughout the structure, providing magnitude and direction data for comparison, internally and externally. Specifically, we validate a FEM of a rat tibia, including trabecular network geometry, using a material testing stage housed within a microCT scanner. We further apply novel landmark based morphometric approaches to more effectively compare modelled and experimental results. 542 landmark points on the cortical and trabecular bone surfaces of the model were selected and validated in 3D against experimental data. This approach may hold considerable potential in fields wherein a better understanding of the mechanical behaviour of trabecular networks is important, e.g., the studies of osteoporosis and trabecular loss after orthopaedic implant insertion.

1. Introduction

Finite Element Analysis is increasingly used to improve our understanding of relationships between form and function in bone and other biological structures (Bourke et al., 2008; Rayfield, 2007; Strait et al., 2009; Wroe et al., 2008). When testing hypotheses using FEA, model validation is commonly an important step to ensure that simulations accurately reflect physical models (Kupczik et al., 2007; Strait et al., 2005).

Validation is typically achieved through the use of strain gauges, which restricts the collection of data to a limited number of external locations (Gray et al., 2008; Strait et al., 2010). A further limitation of this approach is that the strains or deformations that take place internally, e.g., within trabecular networks, cannot be described or analysed. Yet determining the influence of loading on trabecular networks is important for various reasons, including improved understanding of the mechanical consequences of osteoporosis, and trabecular remodelling or loss after orthopaedic implant insertion (Harrison et al., 2008; Harrison and McHugh, 2010; Mc Donnell et al., 2010).

MicroCT scanning is now widely used to study the microstructural and hierarchical properties of bone tissue (Harrison and McHugh, 2010; Muller, 2009). The microCT scanner used in the present study has a material testing stage (MTS) housed inside the scan area. Thus, scanning before and during compression testing can facilitate direct comparisons in 3D between FE simulations and experimental data. Here we take advantage of this capacity to validate a FEM of a rat tibia in 3D. We further apply a new method that utilises morphological landmarks to compare simulated with experimental FEA results.

2. Materials and methods

2.1. MicroCT scanning

A tibia from a Wistar rat was acquired via dissection. The tibial shaft was removed at a point 10 mm below the tibial head. The specimen was then adhered to the base plate using an epoxy resin. We used a SkyScan 1172 microCT scanner with the following parameters: 100 kV source voltage, 10.68 μm resolution, 1180 μs exposure time, and an angular increment of 0.40° about 180° of rotation. The sample was then compressed (0.001 m ms^{-1}) within the elastic region to a load of 12.72 N. A second scan was then undertaken while maintaining this applied load using identical scan settings.

* Correspondence to: School of Biological, Earth and Environmental Sciences (BEES), Biological Sciences Building (D26), University of New South Wales (UNSW), Sydney, NSW 2052, Australia. Tel.: +61 2 9385 3866.
E-mail addresses: samuel.evans@uon.edu.au (S.P. Evans), parr.will@googlemail.com (W.C.H. Parr), Philip.Clausen@newcastle.edu.au (P.D. Clausen), allan.jones@sydney.edu.au (A. Jones), s.wroe@unsw.edu.au (S. Wroe).

0021-9290/$ - see front matter © 2012 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.jbiomech.2012.08.003

2.2. Modelling

The FEM was generated using previously described protocols (Chamoli and Wroe, 2011; Wroe et al., 2008). Surface models, for before and during compression, were created in Mimics (version 14.12), with the 'before model' being imported into 3-Matic (version 3.0) for solid meshing. The resulting FEM, comprising 2,002,478 4-noded tetrahedral elements (Fig. 1), was imported into Strand7 (version 2.4) where constraints and loads were applied to simulate the experimental set-up. Nodes at the distal end of the tibial diaphysis were fixed. A face pressure was applied to the distal region of the tibial diaphysis to represent the epoxy mounting and another surface pressure was applied to the ephyseal articular surface to represent the compression plate.

It was necessary to determine an appropriate value of Young's modulus of elasticity for the sample, using back calculation methods (Arns and Knackstedt, 2002; Tsafnat and Wroe, 2011). An initial value of $E=1000$ MPa was applied to the FE model, which was then scaled to $E=259.8$ MPa, corresponding with the net deformation of the sample and the experimentally applied load. In applying material properties, a homogeneous and isotropic bone simplification was used (Bright and Rayfield, 2011; Nakashige et al., 2011). The model was solved under linear static assumptions within Strand7.

2.3. Landmark based statistical comparison between FEA and microCT results

To compare FEA results with the surface model of the tibia generated from microCT taken during compression, we modified the method of Parr et al. (2012) which used the mean displacement values of the closest 12 FEM nodes to selected *x,y,z* coordinate points. The nearest 12 nodes equate to the nearest three 4-noded tetrahedral elements, i.e. the mean displacement of all of the nodes in the closest three brick elements to each landmark was calculated. Using the mean displacement of 12 nodes minimises single extreme displacement value occurring due to modelling errors that might impact the results. This created homologous landmark points for statistical comparison between the FEM (both before and during

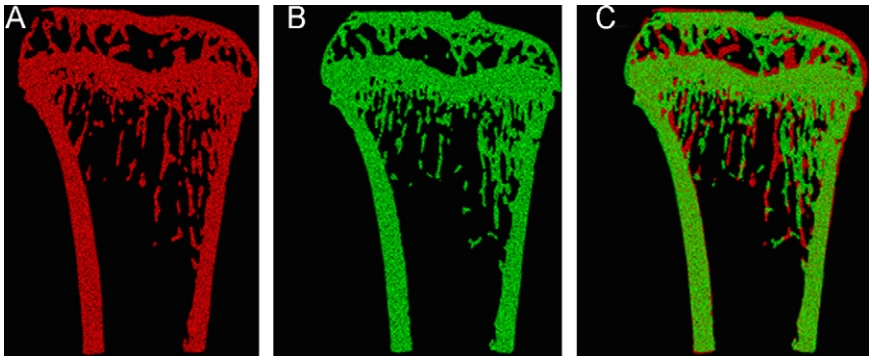

Fig. 1. Sagittal view showing a mesh of the tibia before compression (left), the tibia meshed during compression (centre), and the two meshes overlayed (right). The fact that the two models can be overlayed in 3D and hence compared forms the basis for this methodology.

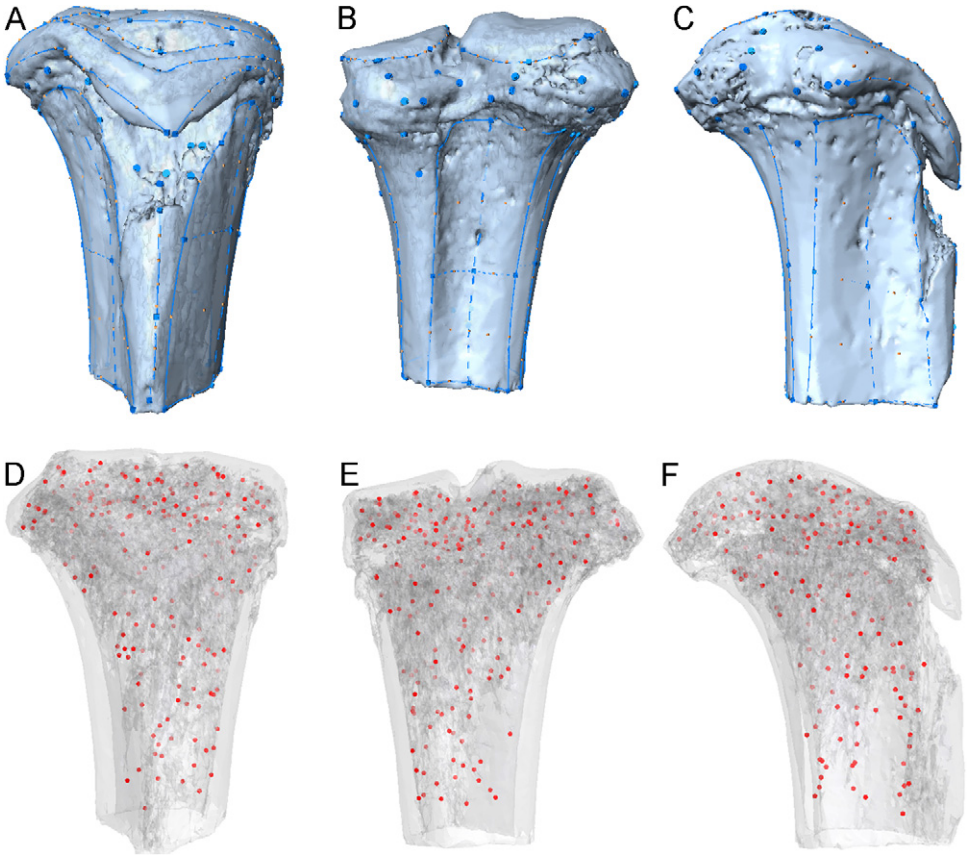

Fig. 2. External cortical, (A–C) and internal trabecular, (D–F) landmark positions. External cortical landmarks anterior (A), posterior (B) and lateral (C) views. Internal trabecular landmarks anterior (D), posterior (E) and lateral (F) views.