SPECIMEN-SPECIFIC FINITE ELEMENT MODELING OF LUMBAR VERTEBRAE: PARAMETRIC STUDY WITH EXPERIMENTAL VERIFICATION

INTRODUCTION
Computational modeling of the spine using the finite element (FE) is becoming widely used in the assessment of new orthopaedic procedures. The continual increase in computational power has led to larger and more complex models being developed. However, the verification of the parameters used in the models has been limited. In particular, little work has been undertaken to verify the modeling of bony failure, although this is important in assessing instrumentation for spinal fracture fixation and vertebroplasty.

The parameters associated with FE models fall into two main categories: those relating to the geometry of the model including the FE mesh and boundary conditions, and those relating to the material properties. Whilst some studies have focused on the assessment of individual parameters such as particular material properties [1] or cortical wall thickness [2], little is known about the relative importance of the two parameter categories.

The aim of this study was therefore to use specimen-specific and generic FE models of porcine vertebral bodies to investigate the effects of both material property and morphological parameters on FE model predictions by direct comparison with experimental data. A further objective was to investigate if such models could adequately predict the extent of plastic damage within the body following load to failure.

METHODS
Experimental tests
Eight lumbar vertebrae were harvested from three porcine spines and imaged using a micro computed tomography (µCT) scanner (MicroCT80, Scanco Medical AG, Switzerland). The bone volume fraction (BVF) was calculated for the central region of trabecular bone using proprietary software. The specimens were then tested to failure under axial compression in a materials testing machine (RDP Howden, Southam, UK). Load-displacement data were taken from the machine and used to determine the failure load and stiffness, which was defined as the gradient of the load-displacement curve over a 1 mm section ending at half the failure load. The height loss of the specimen following loading was also recorded.

Finite element simulation
Specimen-specific FE models of each vertebra were generated from the µCT images using custom-written software (Figure 1). An additional 'generic morphology' mesh was generated from an L3 vertebra not used in the study. For each specimen, three models were used: one using the generic mesh and material properties based on the specimen BVF values (set A: material specific); one using the specimen-specific mesh but generic material properties based on the mean BVF for all specimens (set B: morphology-specific); and one using the specimen-specific mesh and material properties based on the bone area fraction of each element face within the model (set C: material and morphology-specific). In all cases, the bony elements were assumed to behave as a linearly elastic—perfectly plastic solid with both the modulus of elasticity and failure strength proportional to the BVF. Using model set C, the effect of the post-yield behaviour of the bone was further investigated by using an isotropic plastic hardening material property with an increasingly reduced modulus [3].

Each model was simulated under axial compressive load to failure with boundary conditions applied to match the experimental protocol. All of the finite element models were solved using ABAQUS Standard (ABAQUS Inc., Providence, RI). The agreement between the experimental results and those predicted from the FE models were assessed using the concordance coefficient proposed by Lin [4].

RESULTS
From the experimental tests, the specimen stiffness ranged from 3.30 to 6.03 kN/mm (mean = 4.62 kN/mm) and the strength from 6.83 to 11.79 kN/mm. The results of one specimen were discarded due to slippage of the specimen-mount interface during testing.

The concordances between the experimental and FE results are shown in Table 1. The values predicted from the set C models (material and morphology specific) showed higher levels of concordance than either of the less specific model sets for both stiffness and strength and were in good agreement with the experimental results. Although the plastic hardening material model had poorer agreement with the experimental strength measurements than the elastic—perfectly plastic model, it was better able to predict the extent of plastic failure as measured by the specimen height loss (concordance coefficients = 0.869, 0.728 for the plastic hardening, perfectly plastic models respectively).

DISCUSSION
The use of finite element modeling in spinal research is growing, and it is becoming increasingly important to understand how the predicted response of a single model can be extrapolated to a wider patient cohort. The results of this study indicate that morphological parameters play an appreciable role in the model response and that it is therefore necessary to consider the geometry of an individual spinal unit in addition to its material properties. Morphologically-specific models have the advantage that material property values from the image intensity data can be mapped on an element-by-element basis into the model and the greatest agreement with the experimental results was found with this model set.

The results of this study also indicate that a plastic hardening model may be an appropriate method of simulating bony failure. This method has the advantage that the same simulation may be used to model the fracture of the spine and then the reloading following treatment, for example by vertebroplasty or posterior instrumentation.

The range of the experimental data was considerable and much of the variation in response appears to be due to the position at which the posterior elements were removed, since the bone in this region was relatively dense. This has implications for experimental studies where single vertebral bodies are used, and care should be taken in interpreting results where ‘between-specimen’ comparisons are made.

At present, methods of segmentation and mesh generation from image data are still not ideal for spinal FE modeling, particularly for the complex geometry of the posterior elements. If the finite element method is going to be developed as a clinical tool for the assessment of spinal treatments, more automated methods of creating patient-specific models are therefore required.

REFERENCES

Figure 1: µCT image and corresponding FE model of one specimen.