

# CANCELLOUS BONE FRACTURE MODELED BY FINITE ELEMENTS AT MICRO SCALE

Ricardo Belda, Ana Vercher-Martínez, Eugenio Giner

Center of Research in Mechanical Engineering – CIIM, Department of Mechanical Engineering and Materials, Universitat Politècnica de València, Valencia (Spain)

## Introduction

The estimation of failure parameters of bone are clinically relevant as in our aged society bone injuries are having greater economic impact. At the macro scale, bone is mainly composed of cortical and trabecular bone. Trabecular bone has major importance in the failure process in bone. The highly hierarchical structure of bone makes it necessary to study its different scales to insight how the fracture occurs. In this work, cancellous bone fracture in two porcine trabecular vertebrae specimens at the micro scale is modeled by finite elements using a strain-based fracture model. The simulations are correlated with experimental tests. This is a preliminary study whose main aims are the estimation of the failure parameters for the fracture model considered and the fracture plane observed experimentally in a compression test of a trabecular specimen at the micro scale.

## Methods

Two cylindrical trabecular bone cores were harvested from two porcine vertebrae and machined such that the main trabecular orientation was aligned with the axis of the core. The specimens were scanned using micro-CT (SUINSA Medical Systems, Spain) with a voxel resolution of 45 $\mu$ m. The micro-CT images were segmented using an in-house developed methodology in Matlab.

Experimental compression test (MTS Criterion c42) were carried out in both specimens using quasi-static conditions reaching fracture.

Cubic trabecular volumes were selected within the segmented images for the FE fracture analysis. The FE meshes were generated using Simpleware ScanIp software and isotropic elastic material properties were defined for the numerical models. The simulations were carried out to reproduce the compression experimental tests. To calibrate the elastic response of each specimen, an inverse identification procedure was utilized, assigning an isotropic Young Modulus such that the elastic response of the numerical model matches the experimental data.

In order to model the complete fracture of the trabeculae, an element deletion technique was implemented using an Abaqus user subroutine (VUSDFLD), based on maximum principal strain [1].

$$f = \frac{\epsilon_{max,ppal}^k}{\epsilon_c} \quad (1)$$

Any element is deleted from the simulation when the maximum principal strain reaches its critical fracture value.

## Results and discussion

Fig. 1 shows the final step of the compression fracture FE model for specimen 1. It can be noted the oblique fracture plane associated to a shearing fracture mode at the macro scale. At the trabeculae level, different local failure modes (bending, buckling and shearing) can be distinguished.

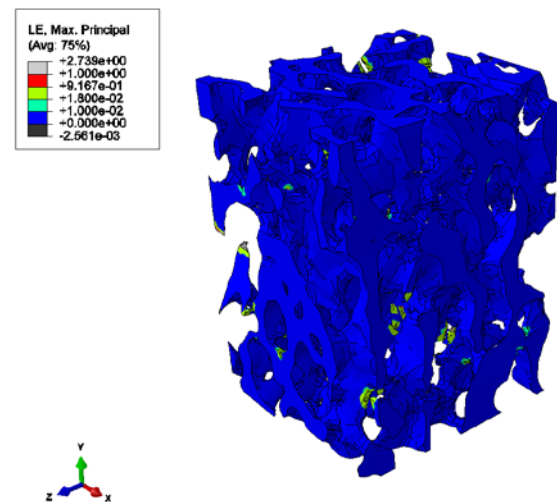


Figure 1: Specimen 1 compression fracture finite element model.

Specimen #	$\epsilon_c$
1	0.028
2	0.032

Table 1: Failure maximum principal strain calibrated for each specimen.

Table 1 shows the calibrated failure maximum principal strain for each specimen that conducts to the fracture state observed experimentally. The fracture path shows a good correlation between simulation and experimental tests. The fracture compression strains obtained for trabecular bone are similar to other values in the literature [2].

## References

1. Keaveny TM, Morgan EF, Niebur GL, Yeh OC. Biomechanics of trabecular bone. *Annu Rev Biomed Eng*, 3:307-33, 2001.
2. Hambli R, Micro-CT finite element model and experimental validation of trabecular bone damage and fracture. *Bone*, 56:363-374, 2013.

## Acknowledgements

The authors wish to thank the Ministerio de Economía y Competitividad for the support received in the framework of the project DPI2013-46641-R and to the Generalitat Valenciana, Programme PROMETEO 2016/007.

