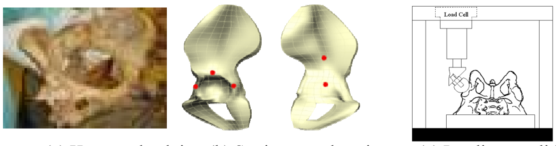
DEVELOPMENT AND VALIDATION OF FINITE ELEMENT (FE) FORMULATION
FOR BOTH LINEAR AND NON-LINEAR CONSTITUTIVE RELATIONS BY
PATIENT-SPECIFIC FE MODELS OF THE PELVIS GENERATED WITH SPARSE DATA SETS

+Shim, V B; **Pitto, R P; *Streicher, R; +Anderson, I A
+Bioengineering Institute, University of Auckland, New Zealand
Vickie Shim: v.shim@auckland.ac.nz

Introduction: Most patient data is limited to a specific region of interest such as the acetabulum, hence cannot be used for FE analysis. We have developed a hybrid method that is capable of generating accurate FE models from such sparse patient data sets. Our formulation uses finite elasticity rather than linear elasticity hence it can handle non-linearity well.

Method: Three pelves were harvested from embalmed bodies. Soft tissues were removed except for the articular cartilage. Five rosette strain gauges were attached to each pelvis around the acetabulum, ilium and ischium (Fig. 1). The loading condition was similar to the one used in [1].



(a) Harvested pelvis (b) Strain gauge locations (c) Loading condition

Figure 1. Mechanical experimental set-ups

FE models were created using the method in [2], which describes how to generate FE meshes from a sparse patient data set. Five different types of mesh per pelvis (total 15) were generated as described in [2] to see how many patient CT slices are needed in generating FE meshes with a good accuracy. For mechanical simulations, we used finite deformation elasticity rather than linear elasticity. This required the use of full Green's strain tensors and a strain energy function in the governing equation. Although computationally more expensive, this setup allowed us to use any constitutive relations. However, the initial validation of this setup was done with linear elastic materials, namely bones. We also used a novel method of assigning material properties, which uses a spatially varying field embedded inside the mesh. Here, each numerical integration point (Gauss point) was given a modulus value specific to its location. Since our FE meshes have large elements due to the use of high order cubic basis functions, the spatially varying field allowed them to have location dependent, inhomogeneous material properties. Contact between the femoral head and the acetabulum was simulated using a contact mechanics penalty method involving Coulomb friction (μ=0.3). The same loading and boundary conditions as the experiment were used in simulation and the results from the experiment and simulation were compared to validate the FE models.

Results: For mesh types 1-4, the number of patient CT slices used in mesh generation varied from 8 to 12 (8,10,11,12) to simulate the situation where only a sparse patient data set is available. Mesh 5 was generated with a full patient data set. All five mesh types showed a good agreement between model and experimental strains, but among Mesh 1-4, the mesh generated with the largest number of patient CT slices (12) always performed the best in predicting strain values.

	Mesh 1 (8 slices)	Mesh 2 (10 slices)	Mesh 2 (11 slices)	Mesh 4 (12 slices)	Mesh 5 (all slices)
Pelvis 1	0.81	0.79	0.80	0.82	0.87
Pelvis 2	0.82	0.76	0.87	0.83	0.83
Pelvis 3	0.81	0.72	0.73	0.79	0.82

Table 1: R^2 values between experimental and FE model principal strains

The von Mises Stress distribution was plotted on the surface of each mesh. A qualitative examination of the surface stress revealed no major difference with the stress distribution obtained from the FE mesh generated purely with patient CT scans (Figure 2).

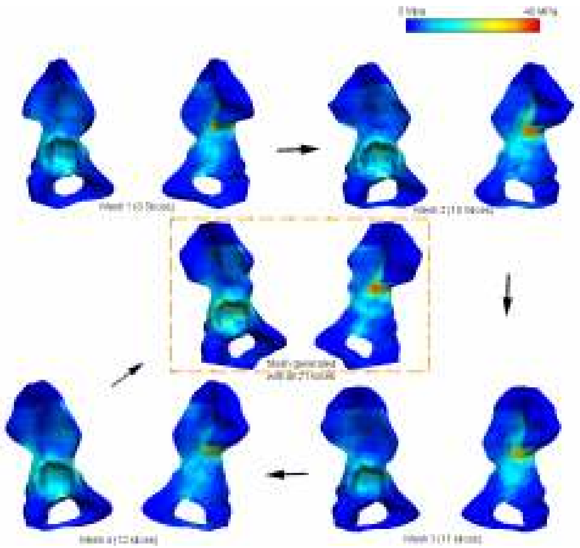


Figure 2: Stress distributions from the five meshes

Discussion: This paper presented the validation of the FE meshes generated with the hybrid method which uses finite elasticity and is capable of generating accurate FE meshes from sparse patient data sets. Comparison of the results from the simulation and mechanical testing has shown that our FE models are capable of producing accurate results. Future works will focus on modelling soft tissues such as ligaments and muscles with our FE formulation.

Reference: [1] Dalstra et al. J. Biomech. Eng., 117:272-278,1995 [2] Shim et al., J. Biomech., ., (PMID:16427645); [3] Dalstra et al. J. Biomech: 26:523-535, 1993;

** Department of Orthopaedic surgery, University of Auckland, New Zealand * Stryker SA. Thalwil, Switzerland