INTRODUCTION:
Finite Element Analysis (FEA) has become an established method for assessing orthopaedic implant designs [1]. Despite interpatient variability in bone quality and morphology having been shown to have a significant influence on the success of joint replacement procedures [2,3,4], the majority of computational studies use only a single bone model for analysis. The greatest number of bone models previously used has been 16, with the analysis results indicating that even this sample size was insufficient to be able to extrapolate results to an entire population [5].

Creating and analyzing an FE model of the femur from medical images using the traditional method has been estimated to take ~8 hours [5]. This manually intensive and time consuming process, together with the limited availability of high quality image data are key factors behind the limited sample size of many computational studies. Addressing this issue, a statistical model of the femur has been developed as a tool for generating a large number of FE ready models [6]. The aim of the current study is to carry out a large scale, multi femur analysis of the influence of femoral head resurfacing on load transfer within the femur. Using the statistical model to generate femurs, resurfacing and subsequent FE analysis of both the intact and implanted scenarios under a walking load will be fully automated. The study replicates the work of Radcliffe et al. [5] and aims to emulate the trends of strain distribution changes seen as a method of validating the method.

METHODS:

260 FE ready femur models, each with unique geometry and material properties, were generated by sampling a statistical model built from 46 Computer Tomography scans [6]. A fully automated methodology was scripted to perform and link all stages of the analysis. Each femur was measured to assess the size and alignment required for the resurfacing component. HyperMesh™ (Altair Engineering Inc, USA) was used to perform Boolean operations, replicating surgical cuts, and to generate a 3mm thick layer simulating bone interdigitated with cement. Ansys ICEM™ (Ansys Inc., Canonsburg, PA) was used to create a tetrahedral mesh of the bone (0.5-1.5mm proximal, 2-4mm distal), cement (0.75-1.5mm) and implant (0.75-1.5mm) with nodal correspondence across their shared boundaries (Figure 1). The elemental material properties of the cut bone were then reassigned from the original femur mesh.

Each femur was run in intact and implanted configurations, with identical subject specific load cases applied simulating a walking load [7]. The load was defined by estimating patient height from femur length, randomly assigning BMI from a distribution curve, leading to a body weight value. A static, linear elastic FE analysis was then run and the resulting elemental strain extracted.

In order to compare the strain distributions between the intact and implanted models, each bone was sectioned into 16 parts along the femoral neck axis, defined when aligning the implant, as illustrated in Figure 3. The strain results from the intact instances were assigned to the implanted bone, thus ensuring identical bone volumes were compared.

RESULTS:

The automated methodology for performing resurfacing implantation on generated femur models was able to carry out the entire process successfully for over 85% of all models attempted. All 260 implanted models, 260 intact models and post processing were performed within 48 hours. Implant positioning and alignment was visually inspected for all models and found to be accurate in every case, as indicated by the 5 models illustrated in Figure 2. As has previously been reported, strain shielding occurs in the most proximal sections of the head (sections 1 and 9). The sections indicated to be most likely to see an increase in strain are those of the proximal femoral neck.

DISCUSSION:

260 femurs were generated, implanted and loaded to simulate the stance phases of level gait in both intact and implanted configurations. The devised implant sizing and alignment methodology was able to reliably position the femoral resurfacing in a suitable orientation, as illustrated in Figure 2. The pattern of change in load transfer through the proximal femur shown by the results of this study match the trends of the previous work by Radcliffe et al. [5], indicating significant strain shielding in the proximal femoral head and an increase in strain in the proximal neck.

This work illustrates the potential application of statistical models as a method of incorporating interpatient variability into computational analysis. In previous work this was used to model only intact femurs [8], however the methodology described in this study shows it is possible to include implants in large scale, population based FE analysis. The possibilities for preclinical implant analysis using the methodology outlined are immense. For the first time it will allow proper statistical analysis to be performed on a representative population.


ACKNOWLEDGEMENTS: Technology Strategy Board, UK.