Young Researcher: Alice Younge
Institution: Imperial College London
Funding body: EPSRC

Supervisor: Dr. Andrew Phillips, andrew.phillips@imperial.ac.uk

Structural Analysis of the Femur: a Collaborative Tool for Surgeons and Engineers

Finite element (FE) modelling has been widely used to create and assess models of the musculoskeletal system. However to achieve a high degree of resolution in describing the structural behaviour of the system, significant computational power and time are required. The objective of this study was to introduce a complimentary approach to FE modelling using a structural approach. This requires far less computational power and models can be analysed in a fraction of a second, offering quick, intuitive results for engineers and surgeons.

The stiffness method was first introduced as an approach for analyzing the behaviour of long bones in 1917. It was used as the *de facto* method for several decades. The introduction of FE modelling offered great advances; beam theory calculations were considered laborious and less accurate. However with the advances in computational power so too comes the ability to create modern automated models analysed using the stiffness method.

A study was conducted using the commercially available general structural analysis software Oasys GSA. A synthetic biomechanical femur was CT (computed tomography) scanned and the solid geometry constructed. This model was sectioned into approximately seventy sections in the regions of the shaft and condyles at the knee, thirty in the neck and thirty in the head. Line plots of the shape of each of the sections, for both cortical and trabecular bone, were then imported into Oasys GSA. The centroids, area, second moments of area and torsion constant were calculated for each section. The sections were plotted at the position of the cortical centroid and parallel axis theorem was used to plot the trabecular section in the same position.

Cable elements with defined force-displacement relationships were used to characterise all muscles and ligaments crossing the hip and knee joints. The model was then subjected to a loading condition representing single leg stance, with a body weight force being applied to the L5S1 joint at the base of the spine. Figure 1 shows comparative graphical views of the stiffness method and finite element models.

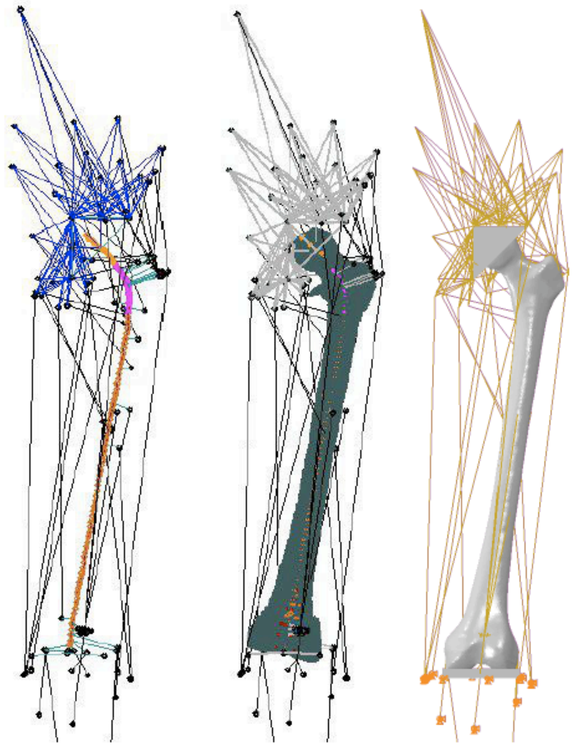

Figure 1: Comparative graphical views of the free boundary condition stiffness model line plot (left), and section plot (middle) and the equivalent finite element model (right).

Oasys GSA produced near instant results showing moment and deflection characteristics of the femur. This data was then used to predict strain plots, which were directly compared to FE results. The results compared very favourably between the two models, as shown in Figure 2.

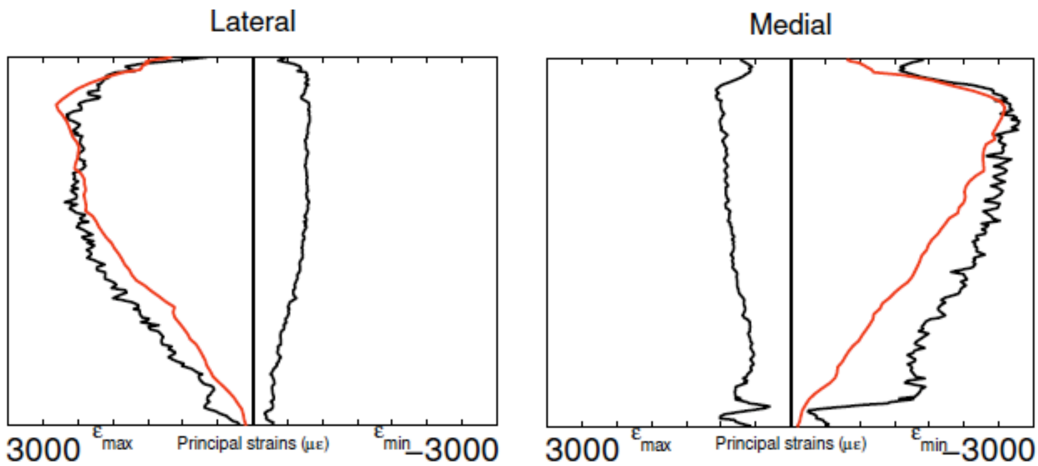

Figure 2: Medial and Lateral principal strains on the surface of the cortical bone comparing finite element (black) and stiffness model (red) results

This study has demonstrated an updated fast, efficient and intuitive alternative to finite element modelling. A significant advantage is that it can be easily interpreted by both engineers and surgeons allowing closer collaboration.