Strathprints Institutional Repository

Gislason, M.K. and Nash, D.H. (2008) *Using finite element method in preoperative planning for wrist surgery.* In: Engineering the Upper Limb: Patient Specific Surgical Planning and Implementation, 2008-04-23 - 2008-04-24, London, UK.

Using Finite Element Method in Preoperative Planning for Wrist Surgery

Magnús K. Gislason[1], David H. Nash[2]

[1] Department of Bioengineering, University of Strathclyde, Glasgow, UK,
magnus.gislason@strath.ac.uk
[2] Department of Mechanical Engineering, University of Strathclyde, Glasgow, UK

Introduction

The wrist has a wide variation of 'normal' anatomy, which may explain the discrepancy seen in success rates of some clinical procedures between patients with similar symptoms. Previously published finite element models of the wrist joint have been based on a single geometry and/or single loading condition which does not give a full representation of the spectrum of normal wrists. In this study, three finite element models of the wrist were created and used subject specific boundary conditions thus building a set of models which can be identified as a part of a larger population. Systematic variations in anatomy and bone position were studied and the effect they have on the general load transfer through the normal wrist joint. That information can prove to be important for future surgical planning on the wrist joint.

Methods

Three subjects were taken for high resolution 3 Tesla MRI scans which were used as the basis for the geometry generation. The in-plane axial resolution of the scans was 230x230μm and a slice thickness of 750μm, a length total of 63.7 mm ranging from the distal end of the radius and ulna to the proximal third of the metacarpals. The scans were imported into Mimics (from Materialize v.9) where the edge detection of the bones and cartilage was performed. A three dimensional object was created of each bone from the contours. A semi automatic meshing procedure within Mimics was used to mesh the bones using triangular surface elements. The surface mesh was imported into Abaqus (v.6.6-1) where a volumetric mesh, using 10 node tetrahedral elements, was created. The material properties of the elements were determined using the grayscale values of the MRI scans. Within Abaqus, the assembly of the bones was conducted. Ligaments were modeled as non-linear spring elements where origin and insertion points were evaluated manually according to previous anatomical studies. Subject specific loading conditions were obtained using series of biomechanical trials. The maximal grip force was measured with the wrist in a neutral position. Five six degree-of-freedom force transducers in a grip device were used in conjunction with VICON motion analysis to define the three-dimensional load systems applied to the hand. Internal loading on the metacarpals was calculated using inverse dynamic techniques. These data were applied as loading conditions in the finite element model to simulate the grip activity. The proximal end of the radius and ulna were held rigid and the load distribution calculated.

Results

The results showed a load distribution ranging from 63.7% to 72.7% going through the radius and 27.4% to 36.2% through the ulna. The overall stress distribution in the carpus varied between the individuals where it is believed that the rotation between the carpal bones has a significant effect on individual bone stress concentration and how the load is transmitted through the carpus. That is directly linked to the intersubject variability and emphasizes how the anatomical variance influences the load transfer.