

ON THE CREATION OF A SUBJECT SPECIFIC FINITE ELEMENT MODEL OF THE WRIST JOINT

Gislason MK.^{1*}, Nash DH.², Barnes R.¹

¹ Department of Bioengineering, University of Strathclyde, Glasgow, UK,

² Department of Mechanical Engineering, University of Strathclyde, Glasgow, UK

*magnus.gislason@strath.ac.uk

INTRODUCTION

Anatomy varies greatly between individuals and therefore it can be inaccurate to derive any clinical conclusions based on a single computer model. It is important to create models which are directly linked to a specific subject who then can be identified as a part of a larger population¹. By these means it is possible to draw conclusions about the discrepancy between two or more subjects or two or more subject groups.

Advances have been made to create a subject specific finite element model of the hip, by using automated procedures². The hip poses a relatively simple geometry for such robust procedures to be implemented. However when faced with a more geometrically complex joint such as the wrist joint or the ankle joint, the procedure becomes more laborious since automatic procedures become impossible to apply. The geometry is the single most important factor for modeling such types of multi-bone systems and there needs to exist a good balance between creation time and level of accuracy and mesh refinement.

In previously reported finite element studies of the wrist joint, ad hoc boundary conditions have been applied to the system. In creating a subject specific model it is important to apply boundary conditions that have been measured from the particular subject. Coupling subject specific boundary conditions with accurate application of material properties of the bones and soft tissues allows the creation of models to predict realistic in-vivo stresses on the carpal bones.

In this study three subject specific finite element models were created of the wrist joint, ranging from the distal end of the radius and ulna to the proximal third of the metacarpals, a total of 14 bones were included in the model.

MATERIALS AND METHODS

High resolution 3 Tesla MRI scans were obtained from the subjects and 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. The image size was 512x512 pixels and the imaging consisted of 92 axially sliced scans, a length total of 63.7 mm. By using MRI scans oppose to CT scans it is possible to visualize soft tissue such as cartilage and tendons which would otherwise be invisible to the CT scan. 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 automatically which was based on the surface mesh. This method has been criticized in the literature, but by running element quality checks then ill shaped elements can be identified. If any badly shaped elements were discovered, then iterative processes was carried out by refining the surface mesh and reconstruct the volume mesh.

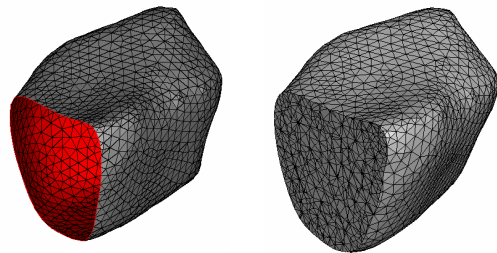


Fig. 1: surface and volumetric mesh for the radius

The volume mesh was then re-imported into Mimics where the greyscale of the scans were used to identify cortical bone, cancellous bone and cartilage and different material properties assigned to the elements. The volume mesh was finally imported into Abaqus where 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. A series of biomechanical trials were conducted to obtain subject specific loads during a whole hand maximal grip activity with the wrist in the 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.

A set of three models were created representing the wrist in a functional neutral position for three different subjects. The three subjects represented the spectrum of the anatomical variance in the wrist suggested by Craigan and Stanley³

RESULTS AND DISCUSSION

The three models were solved using the explicit solver in Abaqus and the reaction forces calculated at the fixed ends of the radius and ulna and compared with the intersegmental joint forces used as the loading condition

at the distal end of the metacarpals. The percentage loading distribution between the radius and ulna was calculated and compared with other previously published findings.

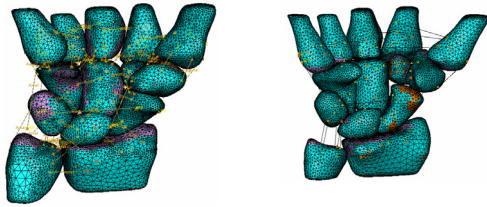


Fig. 2: Finite element meshes for two subjects

The results showed that the load was shared 63.7-72.7% through the radius and 27.3-36.3% through the ulna (table 1). The calculated error between the input forces and the output forces varied between 0.6-4.9%.

	Radius [%]	Ulna [%]	Error [%]
Subject 1	72.7	27.3	0.6
Subject 2	72.5	27.5	1.4
Subject 3	63.7	36.3	4.9

Table 1: Distribution between loading of radius and ulna.

The results show a similar load distribution through the radius and ulna but show a different stress distribution in the other carpal bones whilst looking at the carpus as a whole. The rotation between the bones plays an integral part in how the load is transmitted through the carpus which again is a consequence of the difference between individuals and how their joint position influences the load transfer and stresses the importance of subject specific models.

Parallel and independently carried out cadaver study was performed in the bioengineering unit of Strathclyde University in order to validate the finite element model and provide analysis based on a full three-dimensional representation of the bones of the wrist with subject

specific anatomical and kinetic data. The results are in coherence with the experimental work and support the initial predictions from the model.

A similar methodology of creating such finite element models has been proposed by Cheung et al⁴, who presented a finite element model of the ankle joint where the model was based on a CT scan as opposed to MRI. The benefit of using MRI scans is the visibility of the soft tissue, in particular the cartilage which then can be modeled along with the bones to add to the model's complexity. Using this kind of methodology then the process of creating a subject specific finite element model of a multi bone joint has been made easier, but such joints are too geometrically complex for complete automation to be carried out. This can prove to be important for constructing a reliable method for creating finite element models of such joints in order to be able to model a large population to obtain knowledge of how the bone geometry, positioning and other inter-subject variability can influence the load transfer characteristics.

CONCLUSION

Subject specific anatomical configurations and subject specific joint loading has been used to develop fully representative FE models of the wrist joint. Three subjects with different anatomical configurations of the wrist were studied. The variation of the resulting stress distributions in the bones reinforces the need for subject specific models or a range of models that represent natural variation in the population.

ACKNOWLEDGEMENTS

This work was supported by Arthritis Research Campaign grant no. 1546

REFERENCES

- [1] Taddei *et al.*, J. Biomech, 39:2457–2467, 2006
- [2] Viceconti *et al.*, J. Biomech, 37:1597–1605, 2004.
- [3] Craigen & Stanley, J. Hand Surgery, 20B: 2: 156-170, 1995
- [4] Cheung *et al.*, J Biomech, 38:1045–1054, 2004