FINITE ELEMENT MODELLING OF A TOTAL WRIST IMPLANT

M. K. Gislason*, D.H. Nash
University of Strathclyde, Department of Mechanical and Aerospace Engineering, Glasgow, UK

KEYWORDS
Finite element, biomechanics, orthopaedics, joint replacement

ABSTRACT –
The aim of the project was to create finite element model of a commercially available wrist implant to examine its behaviour under loading. The finite element model was validated by carrying out mechanical tests. Little is known about the mechanics of wrist implants under loading and how they interact with the surrounding bone. Surgical outcomes have been poor and further research is needed to investigate the mechanical behaviour of the wrist implants under loading. A finite element model of the Universal 2 implant (Integra) was created using 3 dimensional scanning of the implant and a CAD model reverse engineered and inserted into a three dimensional model of the radius bone by using Mimics (Materialise). The implant consisted of 3 components: a proximal component made from CoCr alloy, a polyethylene spacer and a titanium distal component. The finite element modelling was carried out in Abaqus (Simulia) and solved using the implicit solver. Loading was applied to the distal component as surface pressure and the proximal end of the radius bone was fixed. Mechanical tests were carried out where the implant was fitted with strain gauges and loaded in uniaxial compression. The results showed good agreement between the FE model and the mechanical tests. Majority of the stresses were transmitted through the proximal stem and areas of reduced loading were seen in the bone. From the results it can be concluded that the design of the implant stem will have great effect on the stress distribution within the bone.

TECHNICAL PAPER -

1. INTRODUCTION

It is estimated that 700 thousand people are suffering from arthritis in the UK causing severe joint pain and reduced quality of life. Each year around 140 thousand hip and knee total joint replacement procedures are carried out in the UK, but only around 200 wrist replacements. The hip and knee prostheses have been developed through a close collaboration between surgeons, academics and industrial partners and numerous studies have been carried out on their mechanical behaviour. Due to little clinical confidence on the mechanical functions of wrist prostheses, surgeons prefer to carry out wrist bone fusion on patients with chronic wrist pain, in order to achieve mechanical stability and reduce pain, rather than carrying out total wrist replacement. Wrist fusion will reduce the range of motion and could have large implications on the quality of life for the patient. A computational study has demonstrated that with bone fusion, the biomechanical behaviour of the wrist bones will drastically change with bone fusion and cause high contact stresses at unaffected joints, leading to degeneration with time (Gislason et al 2012).

Little has been written about the mechanics of wrist implants. In 2004, Shepherd and Johnstone (Shepherd and Johnstone, 2004) proposed a new concept of wrist arthroplasty based on the main principals of the Swanson and the Biaxial designs, using a tapered radial stem in conjunction with a soft flexible rod tying together the proximal and the distal component. Analytical stress analysis was carried out and contact stresses, wear and possible range of motion were reported. The presented design has not become commercially available. However, this is one of the first attempts to quantify the loading through such a device.

In 2004, Grosland et al, created a finite element model of the Universal total implant, and carried out an ex-vivo analysis in order to compare the mechanical behaviour of two different articulation shapes, toroidal and ellipsoidal. The findings suggested that ellipsoidal shape is better suited for the range of motion required of the prosthesis and that the contact area of the toroidal decreases with increased range of motion which could compromise the stability of the prosthesis.

No model exists of wrist implants that predicts the interaction behaviour between the bone and the implant and how the load is transmitted through the radial stem of the implant.

On the market, there exist various different types of wrist prostheses (Universal2, Maestro and RE-motion) which differ significantly in design, where only subtle design changes can be seen in the design of other more established prostheses such as for the hip and the knee. Therefore further studies are needed to assess the mechanical functions of the implants under loading.

In the presented study, a finite element model was created of the Universal 2 implant and results of a simple uniaxial compression compared with experimental data. Then the implant model was then virtually inserted into the radius bone and the load transfer between the implant and the bone investigated.

2. METHODS

The anatomical model of the radius was obtained from subject, scanned using a 7 Tesla clinical scanner. The in-plane resolution was 0.25x0.25mm in plane resolution with a slice thickness of 1mm. The scans were imported into Mimics (Materialise) where the segmentation of the radius was carried out using the masking technique and three dimensional objects created, meshed and exported into Abaqus for analysis (Gislason 10). The 3 components of the Universal 2 implant (radial, polyethylene and distal component) were scanned using industrial scanner at the Advanced Forming Research Centre at the University of Strathclyde, Glasgow, where the geometry of the implant was captured. Each component of the prosthesis was imported as an STL file into Mimics where it was virtually inserted into the radius bone using Boolean operators, thus giving a tight fit between the internal surface of the bone and the implant. The components were meshed in Mimics, using a semi-automatic surface mesher and then exported into Abaqus (v. 6.11) where the surface mesh was converted into volumetric mesh and the finite element analysis was carried out. The mesh consisted of 4 node tetrahedral elements C3D4. The total number of elements was 541,683 for the full model. Interaction between the components was defined either using a surface to surface contact formulation or tie constraints. The connections between the components are listed in Table 1.

Components	Type of contact
Distal part – polyethylene	Tie
Polyethylene – radial part	Surface to surface contact
Radial part – radius bone	Tie

Table 1: Interaction between the components

Loading was applied as uniaxial compressive load to the distal component, simulating compressive forces ranging between 0 and 2000 N (Chadwick and Nicol 2000), which can be expected during gripping motion. Figure 1 shows the finite element models.

Figure 1a – Finite element model of the Universal 2 implant

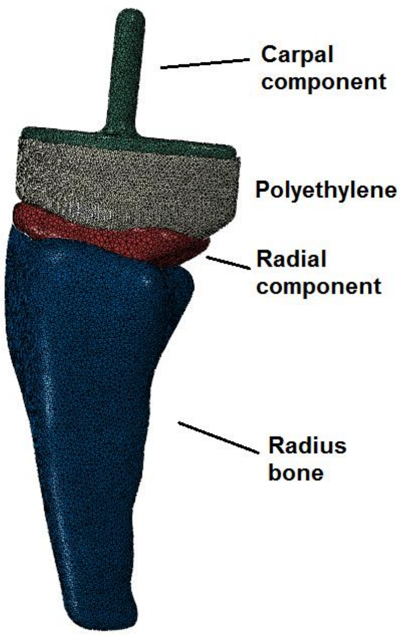

Figure 1b – Prosthesis inserted into the radius bone

The loading was applied as a pressure over the surface of the carpal component. The overall area was calculated as 349.6 mm^2 and a pressure of 5.72 MPa would represent a total load of 2000 N. The proximal end of the radius was kept fixed.

The materials were obtained from the manufacturer. The radial component was manufactured from a cast CoCr alloy (ASTM standard F75, ISO standard 5832-4), the carpal plate component was manufactured from titanium alloy (Ti-6AI-4V ELI, ASTM standard F136, ISO 5832-3) and the polyethylene was made from UHMWPe (ASTM Standard 648, ISO Standard 5834-1 +2). The material properties can be seen in Table 1.

Material	Modulus [GPa]	Yield [MPa]	Tensile strength [MPa]	Elongation [%]
CoCr	207 (220-234)	450	655	8
Titanium	113.8	970	1450	14
Cortical bone	20			
Cancellous bone	0.1			

Table 1: Material properties

The polyethylene was modelled using the Bergstöm-Boyce model (Bergstom and Boyce, 2001) and the material model was obtained from MCalibration (Veryst Engineering) and the model parameters fitted. The reported parameters for the polyethylene model can be seen in Table 2:

μ	λ	D
24.45	1.486	0.004

Table 2: Polyethylene material coefficients

The universal2 implant was mechanical tested where strain gauges were placed on the head of the radial stem of the implant and on the polyethylene component. The implant was tested in uniaxial compression. Figure 2 shows the experimental setup

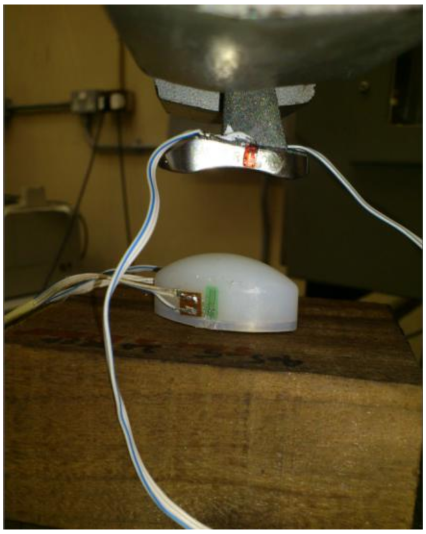

Figure 2a: Experimental setup of the mechanical testing.

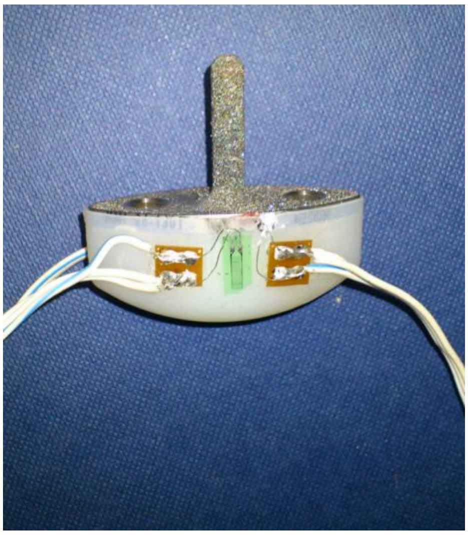

Figure 2b: Strain gauges applied onto the polyethylene component.

The carpal component was glued into wood during the testing to keep it stable and to mimic a material of similar stiffness to bone. Strain data was collected at 100 N intervals with the total force ranging from 0 to 2000 N.

3. RESULTS AND DISCUSSION

The model with and without the bone was solved using the implicit Abaqus solver. First the model was run without the radius bone and then with the bone. Figure 3 shows the stress plot of the bone and implant under compressive loading.

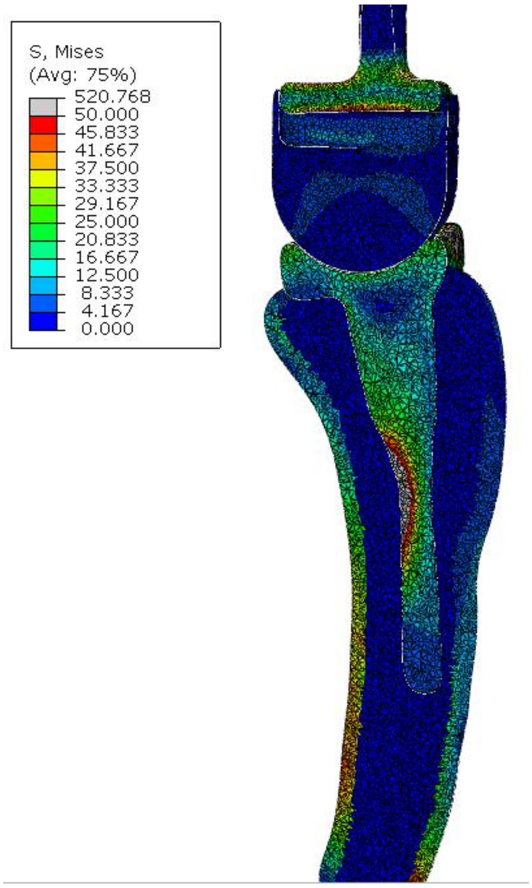

Figure 3a: Stress plot of the radial part of implant and the radius bone.

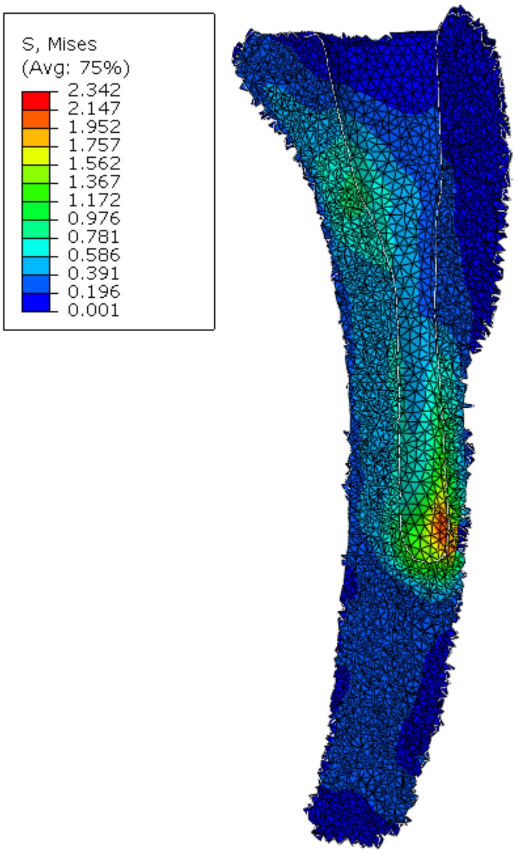

Figure 3b: Stresses on the cancellous bone transmitted from the implant.

From Figure 3a, it can be seen that the majority of loading is transmitted through the cortical shell of the radius and the radial stem, thus unloading the cancellous bone of the radius. Figure 3b shows the stress distribution in the cancellous bone where a region of virtually no stress can be seen on the dorsal aspect. Unloading of the cancellous bone can have serious effects with time as bone that is not mechanically stressed will eventually erode, leaving a gap between the interface between the implant and the bone, which will lead to loosening of the implant and revision surgery.

Another result that can be seen from Figure 3a, are the stresses in the carpal and the radial component of the implant. The maximum stress on the carpal component is 94.5 MPa and the average value is 20.9 MPa. For the radial stem, only 0.33% of the elements exceed values above 100 MPa. The elements that exceed loading of 100 MPa are at the contact surface between the radial stem and the polyethylene and can be discarded as numerical errors. It can be concluded therefore that the loading on the carpal and the radial component is well below the yield limit of the materials. It can be argued therefore whether future stem designs could look into reducing the material in these components, in particular the radial component.

Figure 4a shows the stress distribution in the implant when modelled ex-vivo. The model was constrained by fixing the proximal ends of the radial component and a range of pressure values applied to the carpal plate, replicating the loading conditions in the mechanical testing. The strain values on the polyethylene were calculated and averaged over an area which corresponded to the location of the strain gauges which can be seen in Figure 2b. The measured strain values were compared to the calculated values and plotted as a function of the input load.

The comparison between the test results and the modelling results can be seen in Figure 4b.

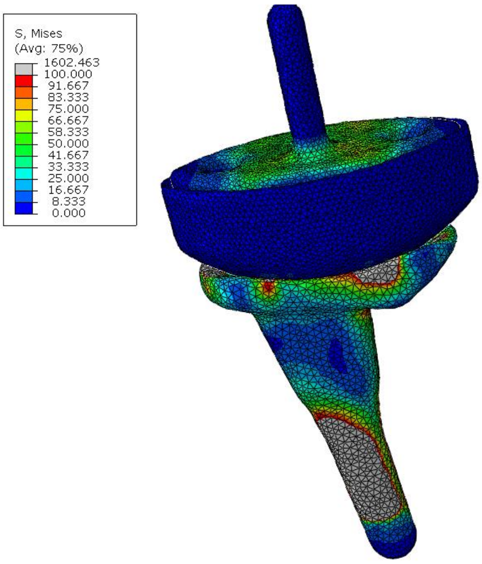

Figure 4a: Stress plot of the implant ex-vivo

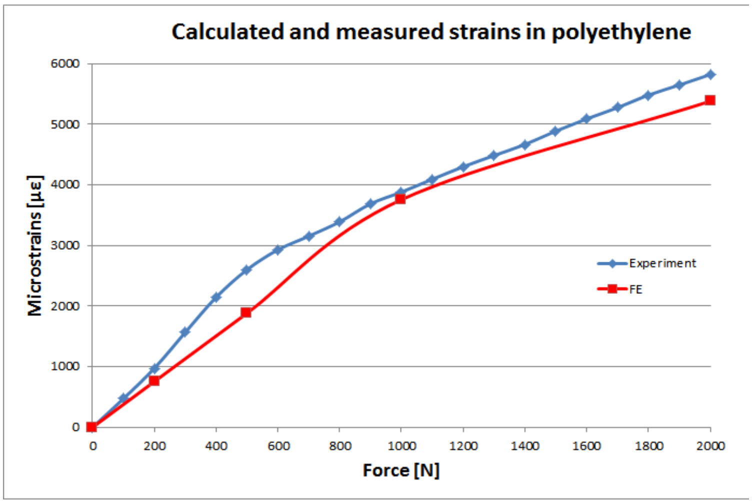

Figure 4b: Comparison between FE model and experimental results

From Figure 4b, it can be seen that the strain values are in agreement although the finite element model seems to be under-predicting the strain values.

Looking at the difference between the ex vivo model and the implanted model, it can be seen that inserting the implant into the radius will greatly affect the stress values as the radial stem is further constrained within the bone than in the ex-vivo model. That emphasises the importance of carrying out the analysis in-vivo.

4. CONCLUSIONS

The paper reports on preliminary findings of the load transfer between a wrist prosthesis and the surrounding bone tissue. From the results it can be concluded that there is scope for re-designing the stem of the implants. Future work would look into measuring the strain values in the radial component inside the radius bone and to create a full implanted finite element model of the wrist.

REFERENCES

(1) Gislason M, et al. J. Hand Surg (Eur), vol 37(9), pp:871-878, 2012
(2) Sheperd, DT and Johnstone AJ. Med Eng. & Phys, 2002, Vol 24, pp: 641-650.
(3) Grosland N, et al. Clin Orthop Rel, Res, No 421, pp: 134-142, 2004
(4) Gislason M et al. Med. Eng. Physics, 32,pp:523-531, 2010
(5) Bergström J and Boyce M, Mech.Materials, 33, pp:523-530, 2001
(6) Chadwick E and Nicol AC, J. Biomech, 33, pp:591-600, 2000