Medical Engineering and Physics 38 (2016) 511–517

Contents lists available at ScienceDirect

Medical Engineering and Physics

journal homepage: www.elsevier.com/locate/medengphy

Mechanical testing and modelling of the Universal 2 implant

M.K. Gislason[a,b,*], E. Foster[b], D. Main[b], G. Fusiek[b], P. Niewczas[b], M. Bransby-Zachary[c], D.H. Nash[b]

[a] *Department of Biomedical Engineering, School of Engineering and Technology, University of Reykjavik, Reykjavik, Iceland*
[b] *Faculty of Engineering, University of Strathclyde, Glasgow, UK*
[c] *Southern General Hospital, Glasgow, UK*

ARTICLE INFO

Article history:
Received 23 February 2015
Revised 18 January 2016
Accepted 5 February 2016

Keywords:
Wrist implants
Finite element modelling
Strain measurements
Mechanical testing

ABSTRACT

Understanding the load mechanics of orthopaedic implants is important to be able to predict their behaviour in-vivo. Much research, both mechanical and clinical, has been carried out on hip and knee implants, but less has been written about the mechanics of wrist implants. In this paper, the load mechanics of the Universal 2 wrist implant have been measured using two types of measuring techniques, strain gauges and Fibre Bragg Grating measurements to measure strains. The results were compared to a finite element model of the implant. The results showed that the computational results were in good agreement with the experimental results. Better understanding of the load mechanics of wrist implants, using models and experimental results can catalyse the development of future generation implants.

1. Introduction

The design of wrist implants has varied greatly in the last decades. The Swanson wrist implant was one of the first implants to gain a commercial success in the US and was based on the concept of a silicone spacer aimed to increase stability in the radiocarpal joint [1]. Fixation was achieved through a proximal radial stem and a distal stem passing through the capitate and into the third metacarpal. With time, a number of fractures on the distal stem were reported [2], leading to a revision of the mechanical design. The next generation implant designs were the Volz [3], and the Meuli [4] implants which demonstrated considerable changes in the overall design compared to the Swanson and used a metal stem made from CoCr and a ball in socket articulations. Other designs followed such as the semi-constrained Trispherical, the Guépar and the biaxial prosthesis which all then were eventually removed from the market [5].

In 2005 Shepherd and Johnston [6] evaluated the design criteria for a total wrist prosthesis in terms of loading conditions, contact stresses, wear rate amongst others. The challenges that engineers face in terms of the overall design of a wrist implant are mainly the small area to fixate the implant components to the bone, in particular in the distal attachment, and the variability of the loads and range of motion. In lower limb implants, such as the knee and the hip, the loading conditions are well defined in terms of gait, ascending, descending stairs etc. The load cases on the upper limb are more ambiguous where gripping, lifting and pushing with the wrist in multiple different positions can occur during activities of daily living. It has been shown that during a key turn action in rheumatoid arthritic patients, the average resultant load on the index finger is 13.9 N [7]. Using a biomechanical model, Fowler and Nicol [7], also calculated the joint contact forces on an MCP implant to find that during the same key turn action, the contact force was on average 182.5 N, which represents the load of a single digit, namely the index finger. Kanellopoulos [8] measured external forces on all fingers during gripping with the wrist in various different positions for young healthy subjects of both sexes and reported that the resultant force on the index finger was on average 23.1 N. It has also been reported [9] that the load distribution between the fingers was in the ratios 35:30:21:14 between the index, long, ring and little finger respectively. Internal loads acting over all five digits were calculated [9] with the wrist in neutral gripping position, using the biomechanical model presented by Fowler and Nicol [7] to find an average resultant force of 1472 N (standard deviation of 320 N) acting on the MCP joints. Chadwick and Nicol [10] reported overall wrist joint reaction forces exceeding 2000 N, during the horizontal power grip in healthy young subjects. Fok and Chou [11] concluded that the joint reaction forces on the MCP joint could be up to 30 times higher than the external forces applied to the fingers. Although a few studies of the biomechanical modelling of the hand exist, there are large variations in the load application to the hand, but all indicate that during gripping the forces through the wrist can

* Corresponding author at: Department of Biomedical Engineering, School of Engineering and Technology, University of Reykjavik, Menntavegur 1, 101 Reykjavik, Iceland. Tel.: +354 599 6344; fax: +354 599 6201.
E-mail address: magnuskg@ru.is, magnuskjartan@hotmail.com (M.K. Gislason).

http://dx.doi.org/10.1016/j.medengphy.2016.02.008
1350-4533/

Table 1
Interaction between the components.

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

be on the order of 1–2 times bodyweight which is a considerably high load given the small size of the joints in the hand and wrist.

Given the success of the hip and knee implants over the years, patient expectations have grown, to have a pain free and a stable joint after total joint arthroplasty. The design and manufacturing process of a joint implant is subjected to vigorous standards from regulating authorities [12]. Development of wrist implants is ongoing and since the introduction of the Swanson implant, many designs have been marketed and some with limited success. Third generation implants like the Universal, the ReMotion and the Maestro are now popular in clinical practice. They all are designed based on using an elliptical contact area between the proximal and the distal aspect which has been shown by Grosland et al. [13] to have an improved property compared to a toroidal shape in terms of stability. All are constructed with a metal stem, a polyethylene spacer and a distal carpal metal plate with screws. Different design aspects can be seen between various implants currently available. This difference can, in particular, be seen between the Universal 2 and the ReMotion on one side and the Maestro on the other. The Universal 2 and the ReMotion have the polyethylene component attached to the distal component in a convex configuration, whereas the Maestro implant has the polyethylene component attached to the proximal stem in a concave configuration, thus resembling more the geometrical features seen in the hip and the knee.

The finite element method is a powerful tool to calculate in vivo stresses on the structural aspect of the human body and has been used with much success to predict loading behaviour on hip and knee implants [14,15]. It is widely used during design processes of various components and can be of great importance for orthopaedic implants where experimental work can be difficult to carry out [16]. Little has been written about modelling of the wrist implants. McCullough [17,18] studied the contact area of various wrist implants under simulated muscle loading from the 3 extensor muscles (extensor carpi radialis longus/radialis brevis/ulnaris) and the 2 flexor muscles (flextor carpi radialis/ulnaris) and demonstrated that the Universal 2 implant had greater contact area than the Biax and the Universal total wrist implant as well as lower maximum stress. Bajuri et al. [19] published one of the first finite element model of the implanted wrist, focussing on the ReMotion implant. Otherwise little has been published on the mechanical properties of the total wrist implants where many studies have looked at the mechanics of total hip and total knee implants.

The presented study, demonstrates validation work carried out on the Universal 2 wrist implant and compares with a finite element model created of the implant. Two different types of strain measurements were carried out, firstly using strain gauges and secondly using Fibre Bragg Grating to measure the strain inside the prosthesis. Fibre Bragg Grating is an established technique in determining strains in various application, given its light weight, flexibility and resistance to corrosion to name a few [20] and has previously been used to measure the strains in bone tissue [21] and contact pressure in total knee arthroplasty [22].

Validation work is difficult to carry out on the wrist joint, as the joint is small and applying measuring devices requires a high degree of joint exposure which will destroy the stabilizing effects of the soft tissue around the joint, thus making it prone to buckling during a uniaxial compression test. In vivo, the load cases on the total wrist implant are more complex than simple uniaxial compression, but by carrying out measurements using a simplified loading scenario and compare to FE model predictions, it will give indications about the mechanical behaviour under more complex load cases. That would be the first step in validating the finite element models. In the presented study, the Universal 2 implant from Integra was used, as it is one of the leading implant used in the UK and the US.

2. Methods

2.1. Finite element model

A Universal 2 wrist implant in size large was obtained. It consisted of 3 components: a radial component, a carpal component and a polyethylene component. All three components were scanned using an industrial scanner at the Advanced Forming Research Centre at the University of Strathclyde in Glasgow where the geometry was reversed engineered into an STL model. The STL geometry was imported into Mimics (Materialise) where the three components manually aligned with each other and virtually inserted into the radius bone. The components were surface meshed using a semi-automated mesher and imported into Abaqus (v.6.11). There the surface meshes were converted into 10 node tetrahedral elements, of type C3D10M. The total number of elements was 246.888 for the full model. The total volume was 10616 mm^3, resulting in element density of 23.3 elements/mm^3. 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.

Loading was applied as uniaxial compressive load to the distal component, simulating compressive forces ranging between 0 and 2000 N [10], which can be expected during gripping motion. Fig. 1 shows the finite element models.

The loading was applied as a pressure over the distal surface of the carpal component. Matlab procedure was written to estimate the surface area by summing up individual areas from each element located at the surface. 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. No slip boundary conditions were applied to the proximal end of the implant.

The materials were obtained from the manufacturer. The radial component was made from a cast CoCr alloy (ASTM standard F75, ISO standard 5832-4), the carpal plate component was made 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 2.

Table 2
Material properties.

Material	Modulus (GPa)	Yield (MPa)	Tensile strength (MPa)	Elongation (%)	Poisson's ratio
CoCr	207 (220–234)	450	655	8	0.31
Titanium	113.8	970	1450	14	0.30
Cortical bone	20				0.2
Cancellous bone	0.1				0.25