

# We are IntechOpen, the world's leading publisher of Open Access books Built by scientists, for scientists

6,900

Open access books available

186,000

International authors and editors

200M

Downloads

Our authors are among the

154

Countries delivered to

TOP 1%

most cited scientists

12.2%

Contributors from top 500 universities



WEB OF SCIENCE™

Selection of our books indexed in the Book Citation Index  
in Web of Science™ Core Collection (BKCI)

Interested in publishing with us?  
Contact [book.department@intechopen.com](mailto:book.department@intechopen.com)

Numbers displayed above are based on latest data collected.  
For more information visit [www.intechopen.com](http://www.intechopen.com)



---

# Finite Element Modelling of a Multi-Bone Joint: The Human Wrist

---

Magnús Kjartan Gíslason and David H. Nash

Additional information is available at the end of the chapter

<http://dx.doi.org/10.5772/50560>

---

## 1. Introduction

Computational models of biomechanical systems have been available for over 40 years. In the first issue of Journal of Biomechanics from 1968 there exists a paper by Marangoni and Glaser looking at the viscoelastic behaviour biological tissue and presented numerical results using a discrete model which can be thought of as a predecessor of the modern finite element models. In 1971 Rybicki et al published a paper on the mechanical stresses of the femur using the finite element method. Since then, published papers on finite element modelling increased yearly and now, 40 years later, the finite element method plays an important part on the analysis of geometrically complex structures. The hip has been researched extensively over these 40 year and numerous papers have been published from various different research groups on the mechanical response of the femur and total hip arthroplasty under various types of loading. What makes the hip an excellent candidate for finite element analysis is the fact that the geometry of the joint is well defined and can be easily extracted from CT or MRI scans but also the fact that the joint contact forces and musculoskeletal modelling of the hip joint has been extensively researched and measured (Bergmann et al 1993) giving a well defined loading condition during gait and other activities. The knee has also been researched using the finite element method where the joint geometry is well defined, but the loading conditions and the kinematics are more complex. Taylor et al (2003) have investigated the performance of total knee replacement using the finite element method.

Modelling of those joints is more complicated than of the hip and knee, due to complex bone geometry, soft tissue modelling as well as difficulty determining the physiologically relevant loading conditions acting on the joint.

The wrist and the ankle pose a challenge in biomechanical modelling due to the complex interactions between the many bones comprising the joint. Each bone will contribute

uniquely to the high range of motion of the joint. The challenge in modelling of the multibone models is to capture the mechanism contributing to the stabilization of the joint. A stable joint is able to provide three-dimensional equilibrium under external loading which can also be interpreted as the ability of a joint to maintain a normal relationship between the articulating bones and soft tissue constraints under physiologic loads throughout the whole range of motion (Garcia-Elias et al. 1995). This implies that the joints need to be capable of distributing loads without generating abnormally high stresses on the articulating surface as well as being able to move within the joint's range of motion. Geometry of the bones also plays an important role in joint stability and the concavity or convexity of the articulating bones helps the bones to distribute stresses across the joint.

Work on finite element modelling of the wrist started in the 1990s with the works of Miyake et al and Anderson and Daniel who modelled the stresses on the radiocarpal joint using a plain strain contact model. That model contained the radius, scaphoid and the lunate as well as the extrinsic ligaments and the scapholunate ligaments. The TFCC was modelled using a series of spring elements. Albeit a two-dimensional model, it marked a beginning of further research interest in the numerical modelling of the wrist. Miyake et al (1994) published around the same time, a finite element model simulating the stress distribution of a malunited Colle's fracture. That same group later published a paper on the stress distribution in the carpus following a lunate ceramic replacement for Kienböck's disease (Oda et al 2000).

Other wrist models were published shortly afterwards and can be summarised in the following table.

Author	Year	Type	Modelled
Miyake et al	1994	Finite element	Radius, scaphoid lunate
Anderson & Daniel	1995, 2005	Finite element	Radius, scaphoid, lunate, ulna
Schuind et al	1995	Rigid body	Whole carpus
Ulrich et al	1999	Finite element	Radius, scaphoid, lunate
Oda et al	2000	Finite element	Whole carpus excluding metacarpals
Carrigan et al	2003	Finite element	Whole carpus excluding metacarpals
Nedoma et al	2003	Mathematical model	Whole carpus
Gislason et al	2009, 2010	Finite element	Whole carpus
Guo et al	2009	Finite element	Whole carpus
Bajuri et al	2012	Finite element	Whole carpus

**Table 1.** Previously published finite element models of the wrist

Carrigan et al published the first three dimensional wrist model where all the carpal bones were incorporated but not the metacarpals. Loading was applied onto the distal aspect of the capitate and was 15 N compressive force which is not representative of physiological in vivo loading on the wrist. Additionally the scaphoid needed to be constrained using unphysiological constraints in order to achieve convergence. In 2009 full three dimensional models of the wrist were published by Gislason et al and Guo et al incorporating the distal ends of the radius and ulna, all the carpal bones as well as the metacarpals. The Gislason model aimed to simulate load transfer behaviour of the wrist during gripping in three different subjects with the wrist in three different positions. The loading was determined on a subject specific basis where the forces and moments acting on the fingers were measured and by using a biomechanical model, the external forces were converted into joint contact forces acting on the metacarpals. The Guo model aimed to simulate the carpal bone behaviour after the transverse carpal ligament had been excised. The loading applied onto the Guo model was a combined 100 N compressive force acting on the the 2<sup>nd</sup> and 3<sup>rd</sup> metacarpal and some unphysiological constraints were applied to the model. Bajuri et al (2012) created a full three dimensional model simulating the effects of rheumatoid arthritis on the stress behaviour of the carpal bones.

Finite element models of the ankle also exist through the research of Chen et al (2003) and Cheung (2004) and although the chapter mainly discusses the creation of a finite element model of the wrist, there are many similarities in the methodology of creating a high quality finite element model of a multi bone joint, whether it be the wrist or the ankle.

The fundamental problems that researchers face in the creation of a finite element model of the wrist are the loading applied and the soft tissue constraints on the carpus. The wrist are a mechanically unstable joint so external constraints, in the form of ligaments, must be applied in order for the carpal bones to return to equilibrium whether they be modelled as spring elements or as separate geometrical entities.

With increased computational power and more enhanced software, it is possible to simulate more detailed structures to a higher degree of detail than before. With the current rate of software and hardware development, the user will soon become the limiting factor on the quality of the finite element models produced.

## 2. Image segmenting

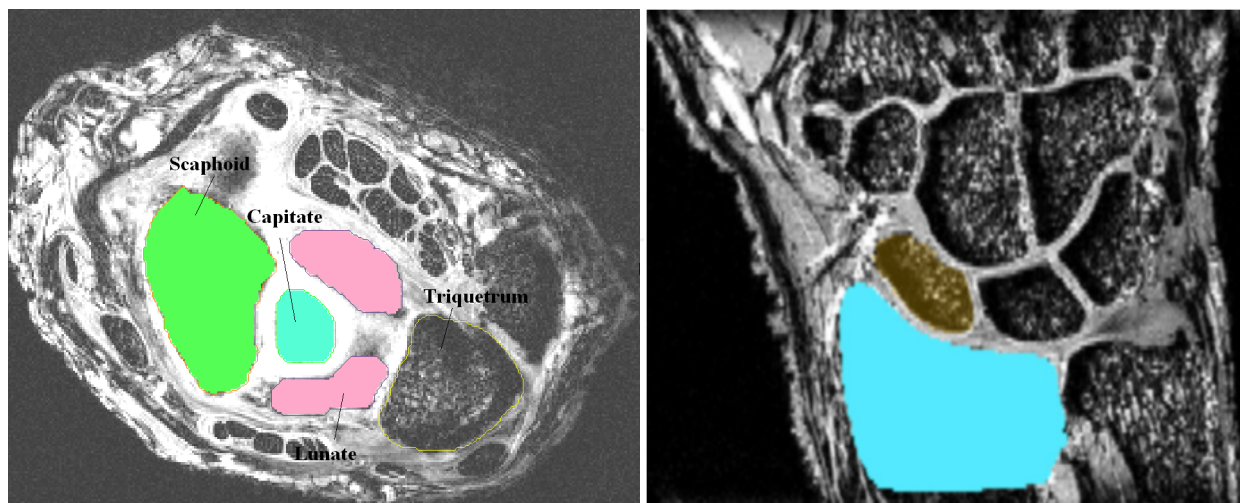
A fully representative geometrical model is integral for the quality of the finite element model. With enhanced scanners and software it is possible to achieve high degree of resolution for the geometrical model. There exist many different image processing software packages that are capable of carrying out image processing and segmenting the scans in order to create three dimensional surface such as Mimics (Materialise), Simplware, Amira, 3D doctor, 3D slicer to name a few.

Segmentation of the wrist bones requires close attention to details as the geometrical features of each carpal bone can be highly irregular and can vary between individuals.

Using an automated segmentation from the abovementioned software packages sometimes can not be enough to capture the full three-dimensional geometry of the bones so manual segmentation is at times necessary. The importance of a high quality segmentation can not be underestimated in multibone modelling as the congruence of the articulating surfaces will play an important role in the contact formulation. Any rough edges on the articulating surfaces will cause penetration of nodepoints causing numerical instability and convergence problems once the finite element model will be run. It is therefore critical to the success of the computational model that the segmentation be carried out in an accurate manner. Another reason why the segmentation is the most critical aspect of the modelling, is the fact that once the geometry has been constructed and meshed, it is very difficult for the user to make any changes to it without starting from the beginning again.

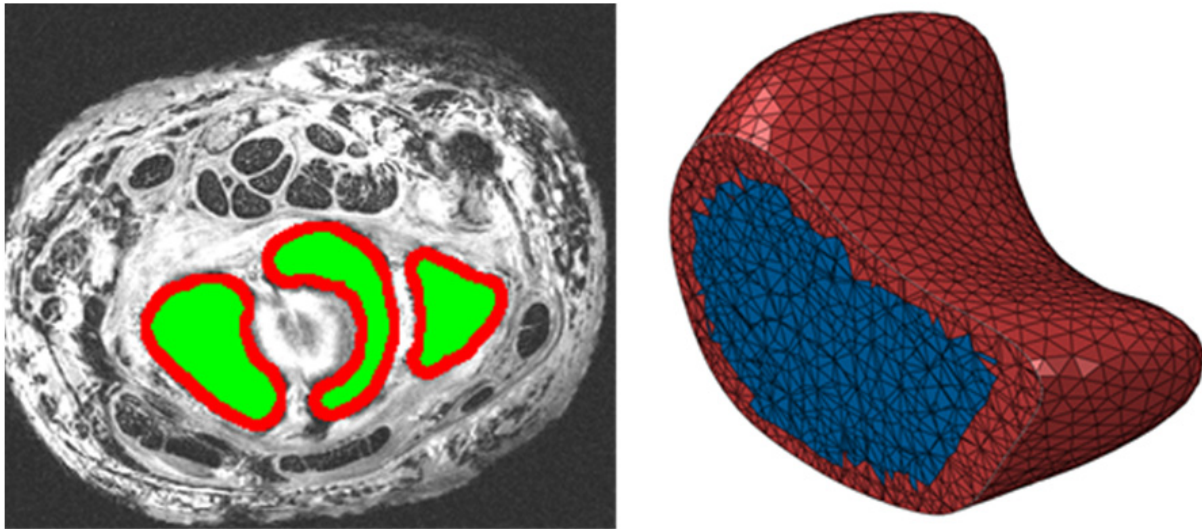
The plane in which the segmentation should be carried out in, would be the plane with the highest resolution, which is primarily the axial plane. Using the sagittal and the coronal plane (or the other two planes with lower resolution) can also be beneficial in order to fine tune the segmentation in order to get a full three dimensional representation of the segmentation. Figure 1 shows segmentation of the carpal bones in axial and coronal planes.

Using the masks can also be a helpful tool in determining the distribution between cortical and cancellous bone. By eroding the mask of a given number of pixels, it is possible to create a hoop in each slice representing the two stiffness layers. Previously published papers have suggested that the thickness of the cortical shell in carpal bones is on average 2.6 mm (Louis et al 1995). Figure 2 shows the distribution between cortical and cancellous bone on the scans and in the finite element model.



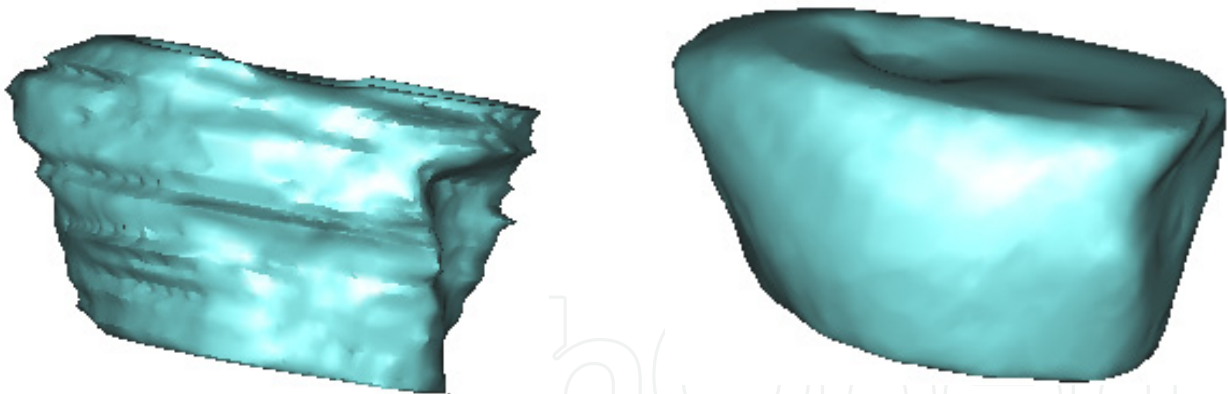
**Figure 1.** Segmentation of carpal bones in two planes





**Figure 2.** Distribution between cortical and cancellous bone in scans and on finite element model

Most software packages now offer the option of smoothing the three dimensional object. It is inevitable that unsmooth edges will occur from the image segmentation and will be more visible if some degree of manual segmentation is required. Figure 3 shows an example of how the radius bone will look like, before and after smoothing.



**Figure 3.** The radius bone before and after three dimensional smoothing

The smoothing is easily done within the software packages, but the user must be aware of the possible implications of the smoothing as it is possible to be too aggressive in the smoothing and therefore Lose volume whilst trying to obtain a good looking picture of the bone. Each iteration of the smoothing causes some changes in the volume of the three dimensional object although some software packages allow to compensate for the volume changes. A possible solution to these volume changes would be to recalculate the mask based upon and carry out manual adjustments of the mask and recalculate the three dimensional object and creating an iterative cycle until the smoothing will have negligible effects on the volume of the bone.

### 3. Meshing

The quality of the mesh of the finite element model will determine the quality of the solution. The process of meshing the three-dimensional objects using an automated meshing tool, which many of the image processing softwares packages discussed in previous chapter have incorporated, has significantly decreased the time and effort to create high quality meshes. The software packages then give the option of importing the meshes into finite element programs such as Ansys, Abaqus and others.

The versatility of the tetrahedral elements have made them popular candidates for the automatic meshing tools in the software packages. The tetrahedral elements are capable of capturing a high degree of geometric non-linearity and are the most popular elements used in biomechanical modelling research today. The problem with the tetrahedral elements is the stiffness of the 4 node tetrahedral element which can give too high stress values compared to the 10 node tetrahedral element. If using a 4 node tetrahedral element, the user must be confident that a sufficient number of elements is being used to capture the nonlinear geometry. For the presented models an average of roughly 430 thousand elements were used, resulting in an element density of about 10 elements/mm<sup>3</sup>.

Hexahedral elements can also be used in biomechanical finite element models. In 2005 Ramos and Simões compared the performance of first and second order hexahedral elements and tetrahedral elements on a femur model and reported that there was little difference in the accuracy of the two types of tetrahedral elements. The tetrahedral elements were closer to a theoretical result, also calculated than the hexahedral elements. The hexahedral elements though showed a higher degree of stability and were less influenced by the number of elements.

As with other finite element models, the mesh quality will play a significant role in the overall solution quality. In a multibody analysis needing contact formulation, obtaining high element quality at the articular surfaces is important, as cartilage elements are soft and tend to deform to a greater extent than the bone elements. Therefore an ill shaped cartilage element, undergoing large deformations, is likely to be excessively distorted and cause divergence of the solution.

With increased computing power, the automatic meshing tool have become extremely powerful and have made it possible that the user will not need to spend much time on producing a high quality mesh, making it possible to model larger numbers of models and incorporating subject specific models.

### 4. Creation of the finite element model

During the creation of the finite element model, the best practice is to import each carpal bone individually allowing the user to keep control over whole assembly. Most of the image processing software packages will take into account the coordinates of individual pixels from the MRI or CT scans. Therefore the position of each carpal bone will be preserved after being imported into the finite element software.

## 4.1. Cartilage modelling

Modelling the cartilage is one of the greatest challenges faced by researchers working on joint modelling. Cartilage is not visible from CT scans, but can be identified using MRI scans. In clinical 3 Tesla scans it can be difficult to determine exactly where the cartilage boundary layer is located in three-dimensional space, making it difficult to create the cartilage layer via masking of the scans. In doing so, the researcher will need to interpolate the shape of the cartilage layer often resulting in an irregular shape causing meshing problems. Another aspect regarding incorporating the cartilage layer into the bone model is the scattering of stiff cortical bone elements and soft cartilage elements. That could cause numerical instabilities in the solution phase. A more practical approach is to extrude the external surfaces of the bones at the articulation and creating a solid volume layer representing the cartilage. Using this method will give a distinct boundary between the bone and the cartilage layer. Another possibility would be to extrude the elements directly creating a layer of wedge elements.

## 4.2. Material modelling

### 4.2.1. Bone

Most finite element models of joints have used elastic material properties for both the cortical shell and for the cancellous bone. Bone is a viscoelastic material and its properties will depend on the strain rate. All published multibone joint finite element models have focussed on a quasi static analysis of the joint and therefore applying the loads slowly. The material properties used for bone material can be seen in Table 2 and are obtained from Rho et al (1997).

Bone type	Young's modulus [MPa]	Poisson's ratio	Density [g/cm <sup>3</sup> ]	Ultimate tensile strength [MPa]
Cortical	17*10 <sup>3</sup> -19*10 <sup>3</sup>	0.25	2000	150
Cancellous	100-200	0.30	1500	20

**Table 2.** Bone material properties

The simplified material values presented in Table 2 will give an idea about the parameters that can be applied to a macroscopical finite element model of a bone. A more refined material model incorporating bone mineral density, the orthotropical behaviour and viscoelastic properties would add a substantial amount of complexity to the model.

### 4.2.2. Cartilage

Many finite element studies have simulated the mechanical properties of the articular cartilage as elastic material which can be subjected to large errors. Articular cartilage is a complex material that has the properties of a fluid and a solid and has been researched extensively in the literature. Much of that research hasn't been applied into the finite element modelling of multibody joints, although many finite element models exist of cartilage only focussing on the material behaviour. Attempts have been made (Gislason 10 and Bajuri 12) to incorporate the non-linearities of the articular cartilage behaviour into the



finite element models, by using Mooney-Rivlin hyper-elastic material properties using the material data obtained from Li et al (2007).

#### 4.2.3. Ligaments

Evaluating the material properties of ligaments pose a great challenge to researchers in multibone joint modelling as they operate only in tension and show viscoelastic material properties. In tension the ligaments show a non-linear characteristic at the initial stages of the load application (usually referred to as the toe region) but once a given reference strain or extension has been exceeded, the ligaments respond in a linear manner to loading. The reason for the non linearities in the toe region is due to the fiber orientation within the ligaments. The collagen fibers are placed in a “wavy” type of fashion and the initial load applied to the ligament goes to straighten the fibers and then they can be stretched in a linear fashion. Another reason is that the fiber lengths within the ligament differ and the initial loading goes to pull the fibers to the same length (Amis 1985). After the linear region then the ligaments follow another period of nonlinear behaviour where the stiffness decreases due to fibre failure until it reaches complete failure

The extrinsic ligaments are generally stiffer but weaker than the intrinsic ligaments which are elastic and strong. In 1991, Logan and Nowak carried out a study where two extrinsic ligaments (the radiocapitate (RC) and the radiolunate (RL)) and two intrinsic ligaments (the scapholunate (SL) and the lunotriquetrum (LT)) were tested to demonstrate the biomechanical difference between the two types of ligaments. Table 3 shows the findings from the study from Logan and Nowak.

Rate	SL [N]	LT [N]	RL [N]	RC [N]
1 mm/min	197.1 ± 35.5	241.1 ± 41.8	50.8 ± 14.8	84.3 ± 16.0
100 mm/min	232.6 ± 10.9	353.7 ± 69.2	107.2 ± 14.5	151.6 ± 23.0

**Table 3.** Results from Logan and Nowak on ligament material properties

From the table it can be seen that the loading rate primarily affects the extrinsic ligaments, making them stiffer and stronger under a rapid loading. This mechanism helps preventing ligament injury during fall, as the extrinsic ligaments anchor the mobile carpal bones to the radius and the ulna.

Tensile experiments on ligaments are difficult to carry out in practice. Wrist ligaments in particular are too short to be tested on their own, so the attaching bones are dissected along with the ligament and are held rigid in the tensile machine. It can be difficult to compare ligament tensile studies because they can be performed under different conditions which can have profound effects on the experimental results on which modellers of the joint rely. Other material studies have been carried out and published in the literature on wrist ligament properties (Berger 1997, Bettinger 1999).

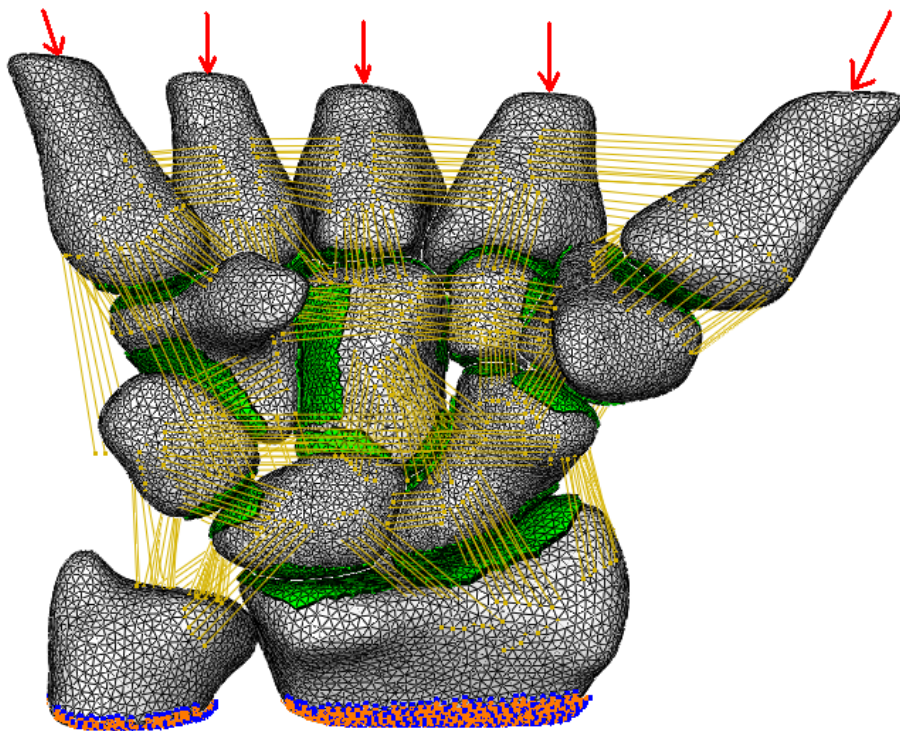
#### 4.2.4. Contact setup

Once the bones have all been incorporated into the finite element software and assembled together bone by bone and cartilage constructed, the contact formulation between the bones needs to be formulated. A surface-to-surface contact is most common method defining the contact between the bones, but node-to-surface configuration can also be implemented. Most finite element models will allow the user different contact models, such as the Lagrange method, the penalty method etc. The availability of these different contact models can be limited to the type of solution algorithm used. Additionally the user can determine the stiffness of the contact, but usually as “hard contact” is applied which is defined by

$$\begin{aligned} p &= 0, & h &< 0 \\ h &= 0, & p &> 0 \end{aligned}$$

Where  $p$  is the contact pressure, and  $h$  is the over closure between the two surfaces. Using kinematic contact method is generally preferred over the penalty contact as it introduces an additional stiffness to the system. Frictionless contact properties or friction using a low friction coefficient should also implemented on the articulating surfaces. By using frictionless contact, it is ensured that no shear stresses occur at the articulations.

It has been reported in the literature (Kauer 1986) that there is little or no movement between certain articulations, such as the articulations between the bones in the distal row of the wrist and the metacarpals (in the carpometacarpal joint). For those joints, it is possible to use a tie constraint so that no relative motion occurs between the two bones. That will help to simplify the model. The model can be seen in Figure 4



**Figure 4.** Finite element model

## 5. Soft tissue modelling

Due to the high mobility of joints such as the wrist and the ankle, they need to be constrained through a large and complicated set of ligaments to ensure structural integrity of the joint. Without any structural contribution from the ligaments, any finite model of the wrist or the ankle would diverge. As previously discussed then the material properties of each ligament will vary depending on its function and location.

The geometry of the wrist ligaments is complex and difficult to incorporate into a finite element model. Some ligaments wrap around the carpal bones without attaching to them, thus providing additional dorsal/volar constraints on the carpus. This can be seen for the dorsal radiotriquetral ligament which originates at the distal end of the radius and attaches to the proximal pole of the triquetrum, overlapping the lunate and adding to the transverse stability of the carpus.

Previous models have incorporated the ligaments as one dimensional spring elements (Carrigan, Gislason, Bajuri), which is the simplest approach of creating the geometry. Although this method will give a relatively good representation regarding the overall constraints of the carpus, the problem will persist that the spring elements will only constrain the carpal bones in the direction of the springs. Using non-linear springs, the user must make sure that the springs do not take any tensile forces. The literature gives a range of ultimate strength and strain values (Berger 1999, Nowak 1991) for various ligaments which can be used to recreate a non-linear stress-strain or force-displacement curve in the form of

$$F = \begin{cases} 0, & x < 0 \\ \frac{\alpha x^2}{2\varepsilon_{ref}}, & 0 \leq x < \varepsilon_{ref} \\ ax + b, & x \geq \varepsilon_{ref} \end{cases}$$

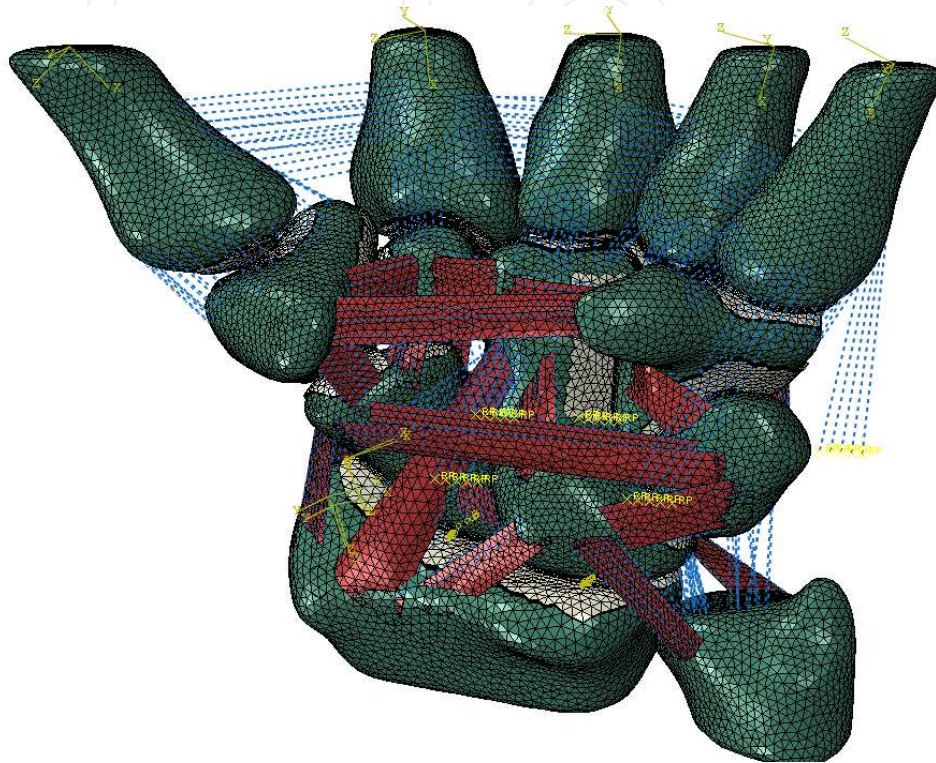
Where  $F$  is the ligament force,  $x$  is the strain and  $\alpha$ ,  $a$  and  $b$  are constants. The force values can be converted into stress, by using measurements of the cross sectional areas of the ligaments as presented by by Feipel et al (1998).

Another possibility is to model the ligament as three dimensional surfaces using two dimensional elements, by identifying the insertion node points and creating the external lines of the ligament using splines, finally an area is defined from the lines and meshed using shell elements. Modelling the material behaviour can be modelled by implementing stress-strain curves for each ligament using hyperelastic material properties. The challenge in soft tissue modelling, beside the geometrical representation of the ligaments, is not over or under constraining the model. A figure of the model where ligaments are represented as three dimensional surfaces can be seen in Figure 5.

In a pilot study carried out on ligament modelling, it was seen that by using the elastic springs, there was a significant translation of the carpal bones, which decreased drastically by assuming linear elastic material properties of the ligaments. That over-constrained the system to a great extent and allowed extremely little bone movement under loading.



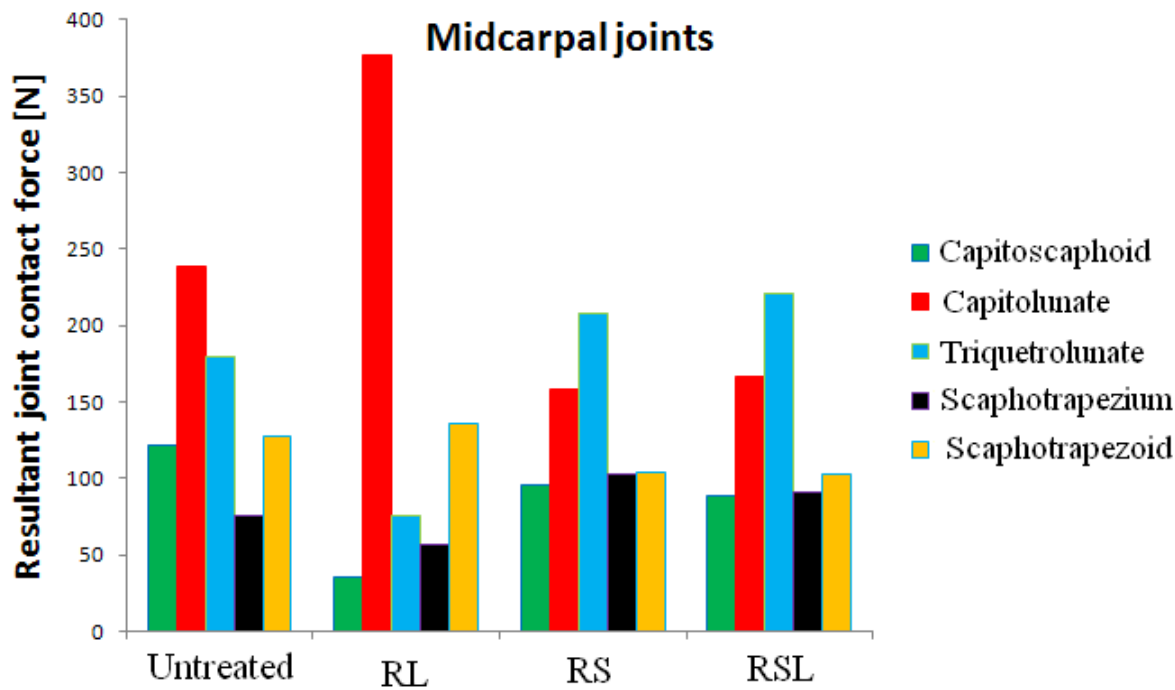
Modelling the ligaments as hyperelastic resulted in larger motion of the ligaments than allowed by using the elastic properties, but less using the non-linear springs. The springs are most probably under- constraining the whole system, but using three dimensional ligaments with elastic material properties are probably over-constraining the system. More research needs to be carried out on the soft tissue properties of multibone joints and the constraining effects various modelling techniques will have on the overall system.



**Figure 5.** Ligaments modelled as three dimensional surfaces.

## 6. Modelling of surgical procedures

With a computational model of the wrist in place, analysis of surgical procedures such as arthrodesis and arthroplasty can be carried out. Arthrodesis is a procedure that fuses together joints to reduce mobility. In the wrist and the ankle there are many individual joints and should just a single joint be fused, the procedure is called partial arthrodesis and a total arthrodesis if the whole joint is fused. This is a recognised surgical procedure to reduce pain and increase stability in the arthritic wrist. Simulating such procedures can be done using a finite element model, where instead of applying contact formulation a tie constraint is applied at the articulating joints. That will treat the two articulating bones as a single unit, not allowing any relative movement between them. After such a procedure it can be seen that the overall load transfer will be altered as additional constraints have been introduced to the system. This can be seen in particular on radiolunate fusion where high joint contact forces were seen on the capitoulunate joint. Figure 6 shows the changes in load transfer in the midcarpal joint following radiolunate (RL), radioscapoid (RS) and radioscapolunate (RSL) fusion compared to the untreated wrist



**Figure 6.** Changes in joint contact forces following a partial wrist arthrodesis.

Using the finite element method can be a useful tool to predict a possible surgical outcome, as can be seen with the radiolunate fusion, an extremely high force can be seen acting on the capitohamate joint. This can be explained by the fact that during gripping (and most other tasks) the thumb will be angled in such a way that the joint contact forces acting on the first carpometacarpal joint will tend to push the carpus ulnarly. This can be seen in Figure 4 how the thumb forces tend to ulnarly translate. With the lunate anchored to the radius and the capitate free to translate, it can be seen that under such ulnarly directed forces the capitate will be excessively constrained by the lunate thus causing such high joint contact forces. It can be seen that by fusing both the radius and the lunate, the model predicts more evenly distributed load through the midcarpal joints, however at the expense of a smaller range of motion.

Finite element models on total hip and knee arthroplasty have been prominent in the literature and extensive research has been carried out on the stress distribution in the femur following a total hip arthroplasty and has contributed to the clinical success of the joint replacements. Little has been written about total wrist arthroplasty and the effects it has on the distribution of load within the wrist. Grosland et al have reported on wrist implants in terms of design and carried out ex-vivo analysis, but a model is missing that captures a full three dimensional features of the implanted wrist. A preliminary model was created of the implanted wrist under physiological loading. It showed how the majority of the load was transmitted through the implant and onto the radius. The finite element model can be seen in Figure 7.



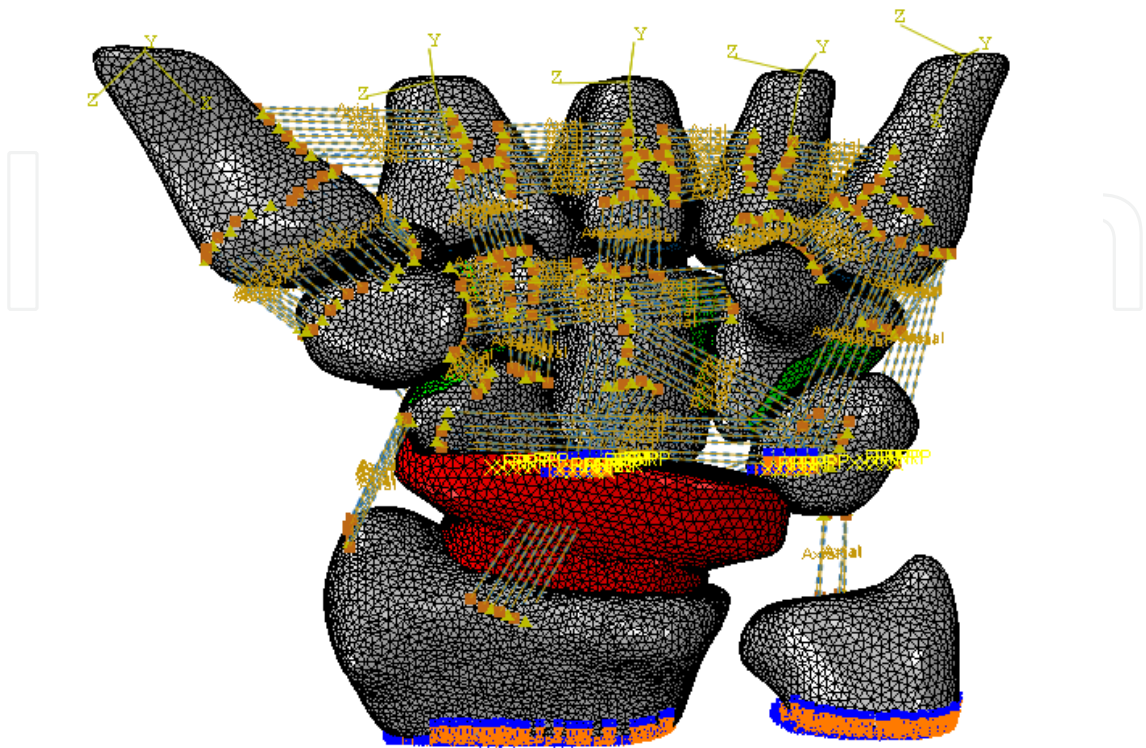


Figure 7. Finite element model of a total wrist arthroplasty

The stresses on the carpal bones and the implant can be seen in Figure 8

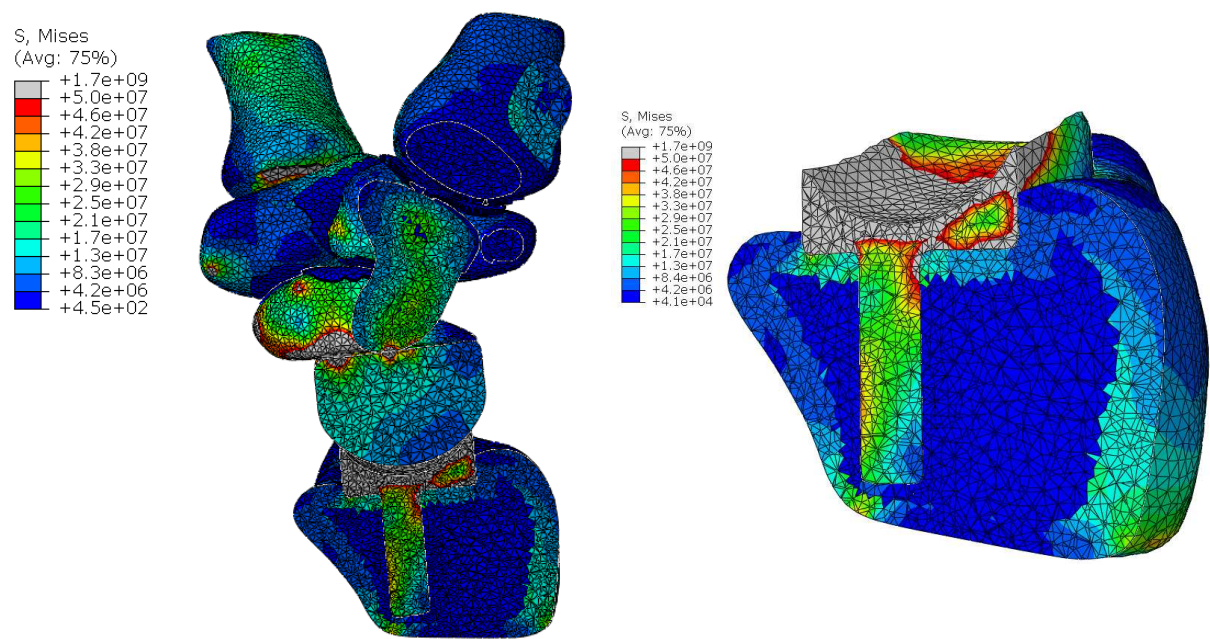


Figure 8. Load transmission through the implanted wrist

From a finite element perspective, modelling a total wrist arthroplasty is a simpler task than modelling the healthy wrist as a few of the carpal bones will be removed during the procedure which will decrease the number of contact surfaces. However problems regarding the fixation of the implant into the radius and the distal row will arise as well as contact between the proximal and distal part. In the pilot study, it was assumed that the implant was fully fixed in the radius as well as the distal component fully tied to the carpal bones in the distal row. There are many different types of wrist implants commercially available and the personal preference of the surgeon will in many cases determine which implant will be used. A finite element model will allow to virtually implant a prosthesis into the carpus and calculate the stresses under static loading. The main problem with carrying out such experiments is that the size and the location of the implant could be erroneous which will have a large impact on the overall solution.

The finite element method can be used as a tool to evaluate the different implant designs available on the market. Given the high failure rate of the implants, there is a demand to investigate closer the effects that a total wrist arthroplasty has on the overall load transfer through the wrist and what can be done to design for longevity and functionality of the implant.

## 7. Loading conditions

Applying in vivo loading conditions on the finite element model, is an extremely challenging aspect of the modelling, especially since there has been very little written about the biomechanical modelling of the wrist. Most studies have applied arbitrary loading conditions, 15 N compressive force acting on the distal end of the capitate (Carrigan et al), a combined compressive load of 100 N applied to the 2<sup>nd</sup> and 3<sup>rd</sup> metacarpal (Guo et al) and a combined 1000 N load acting on the scaphoid and lunate (Ulrich et al). The load cases are better defined when dealing with joints in the lower limb and the fundamental question, researchers must ask themselves is “what activity is characteristic for loading on the upper limb?”. The answer to that is not clear cut and can range from compressive forces acting on the proximal part of the palm with subject trying to push an object to forces action on the fingers via gripping. There are many grip patterns defined in the literature (chuck grip, power grip, pinch grip etc.) which all contribute in a unique manner to the loading distribution through the fingers.

For the analysis presented in this chapter a grip pattern, seen in Figure 9 was used.

The gripping forces were obtained through a biomechanical study where the gripping strength of 50 subjects were measured using five 6-degrees of freedom force transducer (Nano 25-E and Nano 17, ATI Industrial Automation Inc, USA). Simultaneous collection of position data using an 8 camera motion capture system (Vicon, Oxford Metrics Ltd) was carried out to capture both the kinetic and the kinematic data. The external forces were converted in to joint contact forces acting on the metacarpals using a biomechanical model as described by Fowler and Nicol (2000). More detailed analysis on execution of the biomechanical trials can be found in Gislason et al (2009). The wrist models created were subject specific and the joint contact forces applied can be seen in Table 4



**Figure 9.** Grip pattern used for the analysis

	Subject 1			Subject 2			Subject 3		
	Fx [N]	Fy [N]	Fz [N]	Fx [N]	Fy [N]	Fz [N]	Fx [N]	Fy [N]	Fz [N]
Digit 1	144.1	-545.1	-44.6	80.8	-536.1	-8.4	139.7	-452.2	-12.0
Digit 2	253.2	-270.7	141.8	84.1	-294.2	10.5	110.7	-156.8	87.4
Digit 3	348.5	-274.4	172.8	135.1	-126.2	72.8	125.6	-237.7	98.9
Digit 4	117.3	-236.1	29.2	67.0	-94.0	54.7	113.7	-198.0	78.5
Digit 5	111.1	-200.0	-3.8	42.5	-103.0	10.6	53.5	-160.5	19.3

**Table 4.** Internal loading on the digits

Where

- Positive x-direction denotes ulnar direction
- Positive y direction denotes distal direction
- Positive z-direction denotes dorsal direction

As can be seen from Table 4, the contact forces were primarily directed, ulnarly, proximally and dorsally. The joint contact forces were applied to the model as nodal forces where a subset of nodes was chosen and the total force acting on each metacarpal was divided between the nodes.

The proximal ends of the radius and ulna were kept fixed and compressive forces applied to the distal end of the metacarpals.



Many studies have applied arbitrary boundary conditions onto the wrist, which will not give information about the possible in-vivo behaviour of the carpal bones under loading. By applying physiologically relevant loading conditions, it is possible to determine in more detail the mechanical features within the wrist that control the loading. Due to the extensive research carried out on the biomechanics of the hip and knee, modellers are able to apply physiologically relevant loading conditions onto their models and predict in-vivo loading.

## 8. Solution algorithms

For a multibody computational models, it is virtually impossible to solve an implicit model where convergence needs to be obtained for each contact surface for each loadstep. High residual forces at the boundaries of the contact surfaces are primarily seen that cause the solution to diverge. Damping can be introduced between the bones, which can be released gradually as the load step progresses and will be fully released when all of the loading has been applied. Experiments showed that the load step progressed well at the initial stages of the load step, but once the effects of the damping became less, cutbacks were seen in the solution process which increased as the solution reached towards the end of the load step. The solution never will reach the end of the loadstep. This is a classical behaviour of the proper contact not being established between the bones. It has been previously demonstrated in the literature how nonlinearities can cause divergence using the implicit code (Harewood 2007).

Most multibody analyses use the explicit algorithm to solve the model. The explicit algorithm assumes dynamic behaviour of the model and no convergence checks are carried out on the contact surfaces, which makes the explicit algorithm extremely robust in solving such a multi body system. The solution for time step  $t + \Delta t$  is based on the status of the model at the previous time step,  $t$ . In contrast for the implicit code the solution is based on the same time step. The time step in the explicit analysis is determined from the characteristic element length and material properties and is given by

$$\Delta t \leq \frac{2}{\omega_{max}}$$

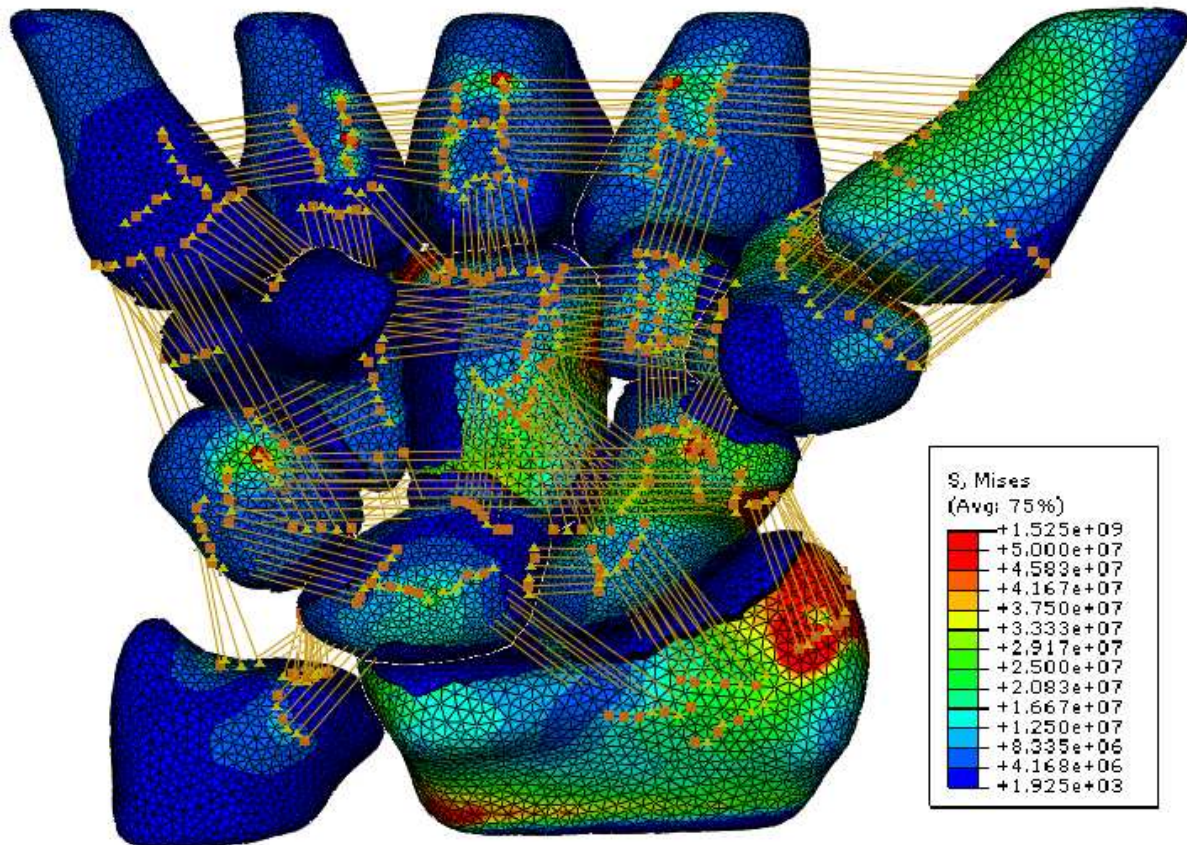
where  $\omega_{max}$  is the maximum eigenvalue in the system. Generally the time steps,  $\Delta t$ , are very small, resulting in long run times. The criteria for assuming a quasi static solution, is that the kinetic energy of the system does not exceed 5% of the strain energy.

## 9. Results

### 9.1. Finite element results

The results from the finite element model have shown that anatomical features play an integral role in the stress distribution through the wrist and therefore it is difficult to generalise about the results of a single standard model. However due to the complexity and time commitment creating the finite element models, it is not possible to generate a large cohort of models.

In the finite element models, the largest stress was seen at the in the cortical shell and were on average a magnitude higher than the stresses in the cancellous bone. On average the stresses in the cortical shell were around 18.6 MPa, and in the cancellous bone they were around 1.1 MPa. The stress distribution for one of the model can be seen in Figure 10



**Figure 10.** von Mises stresses in a single model

Ligaments opposing ulnar translation were more active than others in the model, in particular the dorsal radiotriquetral ligament which showed high degree of force going through it. That result is in agreement with the theoretical findings of Garcia-Elias (1995) who stated that in order to maintain stability, the dorsal radiotriquetral ligament would play an integral part in stabilisation of the carpus during gripping.

The force through the radius and ulna was distributed so that majority of the load was taken by the radius, ranging from 79-93% which is in agreement with the findings of Palmer and Werner (1984) who measured the load distribution between the two forearm bones using a load cell and reported that 80% of the loading was transmitted through the radius.

## 9.2. Validation

Validation is an important procedure to verify that the assumptions used for the computational model are correct. In 2005, guidelines were written by Viceconti regarding the methodology of producing a clinically relevant finite element model. There two

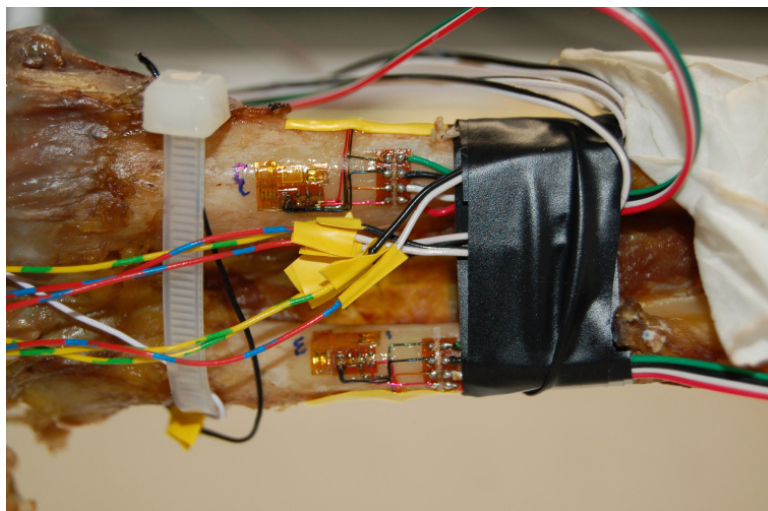


important assessment tools for finite element models were introduced, *verification* and *validation*. The term verification is used to check numerical accuracy, that is how well the underlying equations are solved. To verify the model, the user can check that forces at all reactions sum up to give the input forces. Another example of verification can be seen when energy values are compared to check whether the solution is portraying quasi-static behaviour. The term validation is used to assess how well the underlying equations describe the physical phenomena. Validation must be carried out in the lab to test a specimen under the same conditions used in the computational model. Computational models are capable of creating complex load cases, so through validation some simplification generally must be done, which then can then be re-created through the computational model.

Validation of the computational model was carried out through two separate experiments. One measured the strain on the radius and ulna with the carpus loaded through pull of the tendons (MacLeod 2007). The second measured the joint contact pressure of the radioscaphoid joint using a pressure sensitive film. Setup of the two experiments can be seen in Figure 11.

It was measured using the strain gauges on the radius and ulna that the load through the radius is around 70% and the remaining 30% through the ulna. These values are slightly lower than what the finite element model was predicting, but both recognise the radius as the main load bearing structure of the forearm.

The measurements of the contact pressure on the radioscaphoid joint showed that the joint contact pressure ranged between 4-5 MPa under a 600 N compressive load which is in agreement with the findings of the finite element model which predicted 6.5 MPa contact pressure on the joint under the same loading conditions.



**Figure 11.** Validation of the finite element model

## 10. Conclusions

Creating a finite element model of the wrist and other multibody joints is a complex task where many different aspects of the modelling need to be addressed. The most important aspect contributing to a high quality finite element model is the construction of high integrity geometrical model and the soft tissue modelling. High integrity geometrical model of the articulating surfaces will aid the contact analysis, as a high degree of incongruence of the articulating surfaces can lead to element distortion, especially on soft cartilage elements. The external soft tissue constraints are important in order to maintain mechanical equilibrium as well as allowing the bones to translate and rotate under loading. These two factors will play an integral role in the success of the finite element model.

Finite element models of such complex joints such as the wrist and the ankle are likely to become more prominent in the future as computational power and modelling software quality increases. That will make modellers able to create models incorporation a higher degree of detail than previously has been published.

It is inevitable that errors are introduced in such complex models. The errors can either be within the control of the modeller or without. This chapter has discussed the procedures that the modeller can carry out to minimise the sources of errors in the model. However the modeller will have little control over errors that can be generated through using previously published material properties and geometrical representation of the ligaments and soft tissue.

Using the finite element method predicting the load transfer through the healthy and the pathological wrist can give clinicians important information regarding the choice of treatment which can lead to higher procedure success rates and improve the quality of life for many patients.

## Author details

Magnús Kjartan Gíslason and David H. Nash

*Department of Mechanical and Aerospace Engineering, University of Strathclyde, Glasgow, UK*

## 11. References

- [1] Maragoni RD and Glaser AA: Viscoelastic properties of soft tissue model characterization, *J Biomech*, 1968, 1(1): 33-36.
- [2] Rybicki EF, Simonen FA, Weis Jr EB: On the mathematical analysis of stress in human femur, *J Biomech*, 1972, 5(2):203-215.
- [3] Bergmann G, Graichen F, Rohlmann A: Hip joint loading during walking and running, measured in two patients. *J Biomech*, 1993, 26(8):969-90.
- [4] Taylor M, Barrett, DS: Explicit finite element simulation of eccentric loading in total knee replacement. *Clinical Orthopaedics and Related Research*, 2003, 414:162-171.

- [5] Garcia-Elias M, Ribe M, Rodriguez J, Cost J, and Casas J: Influence of joint laxity on scaphoid kinematics. *Journal of Hand Surgery*, 1995, 20B(3):379–382.
- [6] Anderson DD and Daniel TE: A Contact-Coupled Finite Element Analysis of the Radiocarpal Joint, *Seminars in Arthroplasty*, 1995, 6(1), 30-36.
- [7] Miyake T, Hashizume H, Inoue H, Shia Q, Nagayama N: Malunited Colles' fracture Analysis of stress distribution, *Journal of Hand Surgery (European volume)*, 1994, 19(6): 737-742.
- [8] Anderson DD, Deshpande BR, Daniel TE and Baratz ME: A Three-Dimensional Finite Element Model of the Radiocarpal Joint: Distal Radius Fracture Step-off and Stress Transfer, *The Iowa Orthopaedic Journal*, 2005, 25, 108-117.
- [9] Schuind F, Cooney WP, Linscheid RL, An KN and Chao EYS: Force and Pressure Transmission Through the Normal Wrist: A Theoretical Two-Dimensional Study in the Posteroanterior Plane, *Journal of Biomechanics*, 1995, 28(5), 587-601.
- [10] Ulrich D, van Rietbergen B, Laib A and Rueggsegger P: Load transfer analysis of the distal radius from in-vivo high resolution ct-imaging. *Journal of Biomechanics*, 1999, 32:821–828.
- [11] Oda M, Hashizume H, Miyake T, Inoue H and Nagayama N. A stress distribution analysis of a ceramic lunate replacement for Kienbock's disease. *Journal of Hand Surgery (British and European Volume)*, 2000, 25B(5):429–498.
- [12] Carrigan SD, Whiteside RA, Pichora DR and Small CF: Development of a Three Dimensional Finite Element Model for Carpal Load Transmission in a Static Neutral Posture, *Annals of Biomedical Engineering*, 2003, 31, 718–725.
- [13] Nedoma J, Klézl Z, Fousek J, Kestřánek Z, Stehlík J: Numerical Simulation of Some Biomechanical Problems, *Mathematics and Computers in Simulation*, 2003, 61, 283-295.
- [14] Gislason M, Nash DH, Nicol A, Kanellopoulos A, Bransby-Zachary M, Hems TEJ, Condon B and Stansfield B.: A Three Dimensional Finite element Model of Maximal Grip Loading in the Human Wrist, *Proc. IMechE Part H, Engineering in Medicine*, 2009, Vol 223 (H7), 849-862.
- [15] Gislason M, Stansfield B and Nash D: Finite element creation and stability considerations of complex biological articulations: The human wrist joint, *Medical Engineering and Physics*, 2010, 32:523-531.
- [16] Guo X, Fan Y and Li ZM: Effects of Dividing the Transverse Carpal Ligament on the Mechanical Behaviour of the Carpal Bones under Axial Compressive Load: A Finite Element Study, *Medical Engineering & Physics*, 2009, 31, 188-194.
- [17] Bajuria MN, Mohammed Rafiq Abdul Kadir, Murali Malliga Ramanb, Kamarul T: Mechanical and functional assessment of the wrist affected by rheumatoid arthritis: A finite element analysis, *Medical Engineering and Physics*, 2012, in press.
- [18] Chen WP, Ju CW and Tang FT: Effect of Total Contact Insoles on the Plantar Stress Redistribution: A Finite Element Analysis, *Clinical Biomechanics*, 2003, 18, 17-24.

- [19] Cheung JTM, Zhang M, Leung AKL and Fan YB: Three Dimensional Finite Element Analysis of the Foot During Standing: A Material Sensitive Study, *Journal of Biomechanics*, 2004, 38, 1045-1054.
- [20] Louis O, Willnecker J, Soykens S, Van den Winkel P and Osteaux M: Cortical Thickness Assessed by Peripheral Quantitative Computed Tomography: Accuracy Evaluated on Radius Specimens, *Osteoporosis International*, 1995, 5, 446-449.
- [21] Ramos A and Simões J. Tetrahedral versus hexahedral finite elements in numerical modelling of the proximal femur. *Medical Engineering & Physics*, 2006, 28(9):916-924.
- [22] Rho, JY, Tsui TY and Pharr GM: Elastic properties of human cortical and trabecular lamellar bone measured by nanoindentation, *Biomaterials*, 1997, 18(20), 1325- 1330.
- [23] Li Z, Kim JE, Davidson JS, Etheridge BS, Alonso JE and Eberhardt AW: Biomechanical Response of the Pubic Symphysis in Lateral Pelvic Impacts: A Finite Element Study, *Journal of Biomechanics* 2007, 40, 2758-2766.
- [24] A. Amis. *Ligament Injuries and Their Treatment*, chapter *Biomechanics of Ligaments*, 1985: 3–28.
- [25] Logan S and Nowak M: Distinguishing biomechanical properties and intrinsic and extrinsic human wrist ligaments. *Journal of Biomechanical Engineering*, 1991, 113(1):85–93.
- [26] R. Berger. The ligaments of the wrist. *Hand Clinics*, 1997, 13(1):63–82.
- [27] Bettinger PC, Linscheid, RL, Berger RA, Cooney WP and An KN: An anatomic study of the stabilizing ligaments of the trapezium and trapeziometacarpal joint. *J. Hand Surg. Am.*, 1999, 24(4), 786–798
- [28] Berger R, Imeada T, Berglund L and An K: Constraint and material properties of the subregions of the scapholunate interosseous ligament. *J Hand Surg Am*, 24(5):953–62, 1999.
- [29] Nowak M: *Biomechanics of the Wrist Joint*, chapter *Material Properties of Ligaments*. Springer Verlag New York, 1991.
- [30] Kauer JM: The mechanism of the carpal joint, *Clinical Orthopaedics and Related Research*, 1986, 202, 16–26.
- [31] Feipel V, Salvia P and Rooze M: A new method for measuring wrist joint ligament length changes during sagittal and frontal motion, *Clinical Biomechanics*, 1998, 13(2): 128-137.
- [32] Grosland N, Rogge RD and Adams BD: Influence of articular geometry on prosthetic wrist stability, *Clinical Orthopaedics and Related Research*, 2004, 421:134-142.
- [33] Fowler NK and Nicol AC: Interphalangeal Joint and Tendon Forces: Normal Model and Biomechanical Consequences of Surgical Reconstruction, *Journal of Biomechanics*, 2000, 33, 1055-1062.
- [34] Harewood FJ and McHugh PE: Comparison of the Implicit and Explicit Finite Element Methods Using Crystal Plasticity, *Computational Materials Science*, 2007, 39, 481-494.

- [35] Garcia-Elias M: Kinetic analysis of carpal stability during grip. *Hand Clinics*, 1997, 13(1):151–158.
- [36] Palmer A and Werner F. Biomechanics of the distal radioulnar joint. *Clin. Orthop. Rel. Res*, 1984, 187:26–35.
- [37] Viceconti M, Olsen S, Nolte L and Burton K: Extracting clinically relevant data from finite element simulations (editorial). *Clinical Biomechanics*, 2005, 20:451–454.
- [38] Macleod NA, Nash DH, Stansfield BW, Bransby-Zachary M and Hems T: Cadaveric Analysis of the Wrist and Forearm Load Distribution for Finite Element Validation, Proceedings of the 6th International Hand and Wrist Biomechanics Symposium, Tainan, Taiwan 2007.