Finite Element Modelling of a Human Femur

Final Report

Sarah J. McIlwain[1]

University of New South Wales at Canberra

Medical practitioner and researchers have recently acknowledged the benefit of using Finite Element Analysis (FEA) tools to investigate the stressful effects that current orthopaedic procedures could have upon load bearing structures within the human body. FEA is a cost effective and flexible method of analysing the effects of static and dynamic loads upon one of the most common fracture locations within the body; the human femur [8]. In order to facilitate efficient analysis, a virtual femoral model has been constructed using the ANSYS finite element modelling software. The model's mechanical properties and ultimate strengths have been verified using data gained from both physical composite bone testing and the preliminary literature review. Additionally, conclusions about the necessity of modelling thigh muscles and ligaments in order to improve numerical accuracy were drawn based upon factors of safety associated with static testing of maximal dynamic loads.

Nomenclature

FEA	= Finite Element Analysis	
FEM	= Finite Element Modelling or Model	
SBLT	= Sub Lieutenant	
E	= Young's Modulus in the isotropic plane	$(\mathrm{N/m^2})$
P	= Load	N
δ	= Deflection	mm
σ_{uc}	= Compressive Ultimate Strength	MPa

Contents

---

[1] PLTOFF, School of Engineering & Information Technology

I. Introduction

As the femur, 'provides weight support to the skeletal structure at an up-right position, and is of particular interest in the field of orthopaedics due to being a common fracture site,' [8], a considerable amount of study has been conducted on femurs over the last century, culminating in the current use of FEMs to accurately predict responses to static and dynamic loading [5]. In order to research the effect of post-fracture orthopaedic reconstruction solutions, The University of New South Wales at Canberra, in collaboration with The Canberra Hospital, have facilitated the building of a virtual femur model using the commericial ANSYS finite element analysis package. Currently, many hospitals and research units utilize only physical testing which requires intact cadaveric specimens, surgical procedures to place data probes against a living bone or the use of specially made composite bones which mimic individual femur geometries. Comparatively, a valid FEM has considerable flexibility at a minor cost and can be utilized to analyse any structural situation within a short period of time.

The aim of this project is to create a valid and realistic FEM of the human femur which successfully represents the effect of static and dynamic loading from both stationary and active motions such as standing and walking. The processes involved in this venture include analysing femur geometry and the mechanical properties of cortical and cancellous bone, as well as the application of loads and bending moments in static and dynamic conditions. The ultimate objective of the project is to have an accurate virtual model of the thigh bone which is representative in a wide range of realistic loadings and configurations in order to understand the effects of post-fracture orthopaedic procedures.

The work conducted in 2014 was the second iteration of FEM on the human femur at The University of New South Wales at Canberra. In 2013, SBLT Rebecca Holmes conducted physical testing to verify literary thigh bone mechanical properties and built a virtual femur using the ANSYS FEA software and a 3D femur representation bought from composite bone suppliers, Sawbones. The groundwork and research conducted by SBLT Holmes is still relevant to the second iteration of the femur project and was utilized in verifying the thigh bone FEM. The difficulties SBLT Holmes encountered while creating a model and the solutions found over the course of 2014 will be discussed holistically throughout.

II. Initial Findings

A. Geometry

The human femur is made up of three segments; proximal, shaft and distal [8]. Proximal, meaning situated near the head of the body [13], accounts for the femur head, neck and trochanters seen in Fig. 1. The overall load on the femur is a function of the inclination angle of the femoral head to the shaft, making the neck segment one of the likeliest fracture locations on the femur [7]. The shaft is the long cylindrical continuum between the proximal and distal sections. At the bottom of the femur, producing half the knee, are the condyle and the intercondyle fossa sections which make up the distal end, meaning situated away from the point of attachment [14]. While a large range of axis systems are associated with the femur, this project will utilize the shaft-based system. The shaft body comprises the x axis, the circumferential or transverse direction the y axis and the radial or endosteal-periosteal direction, the z axis [11].

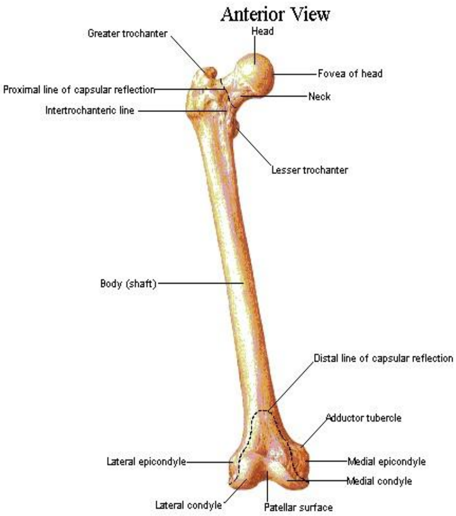

Figure 1: Anterior View of Femur ([11])

Cortical and Cancellous Bone

Two types of bone material exist within the femur, cortical or compact bone and cancellous or trabecular bone [8]. Cortical bone is a very dense, hard material with the largest apparent strength properties, while cancellous bone is a spongy material of irregular latticework approximately 200 μm in thickness [8]. A significant amount of study has been conducted on the mechanical properties of cortical bone and to a lesser extent cancellous, however the wide range of results makes clear consensus difficult; anywhere from 10 to 20 GPa for cortical and 0.05 to 0.4 GPa for cancellous [10], [11].The variety of results are due to a number of issues, including unwarranted mechanical property simplifications [10] and the inherent differences in femur properties due to age, sex, health and race [11]. The individuality in thigh bone

properties cannot be easily mapped and so must be accounted for in statement and by using the statistical mean of a large sample size [4].

This project is modelling an isotropic composite femur instead of a cadaveric bone. The physical composite bone mimics both the cortical and cancellous properties of a cadaveric femur [4] with the basic material properties are available from the composite bone supplier, Sawbones. The cortical bone is simulated using pressure moulded short-glass fibre reinforced epoxy with a compressive modulus of 16.7 GPa and ultimate strength of 157 MPa. The cancellous bone is represented with rigid polyurethane foam [3] and a Young's Modulus of 155 MPa. The ultimate compressive strength of the foam is 6.0 MPa. The Sawbones femur tested was a 4th generation composite bone, based upon the left femur of a 1.83m tall man of 890N weight [12].

B. 3D Femur Geometry

The Sawbones 3D femur geometry that SBLT Holmes used for her FEM was initially going to be utilized in the second iteration of the femur modelling project, however there were a number of issues with the 3D CAD geometry, which precluded it from further use (Fig. 3). The first was the inability to alter the geometric set of the bone model, as Sawbones appeared to have saved the model as a single part, instead of an assembly. As only the faces were accessible, adding cancellous bone or accurately applying loads and constraints was very difficult. The second issues was the hollow nature of the model, which represented only cortical bone with no thought for the significance of cancellous bone in distributing and buttressing against loads [10]. The third difficulty was the large number of sharp edges and discontinuities present within the geometric representation, which heavily affected the meshing capability of ANSYS. SBLT Holmes took steps to improve the geometry using CATIA and ANSYS extrusion and heal functions, however not all of the discontinuities could be resolved and so the initial model was eventually used, with an acknowledgement to its flawed state. The stresses produced were of a magnitude difference to those seen in physical testings which required SBLT Holmes to increase the Young's Modulus to an improbable level in order to obtain realistic results. In her conclusions Holmes' wrote, 'The extreme values needed to represent any strain or displacement correlation indicate that the material properties are incorrect and that poor assumptions were made in the model construction.'

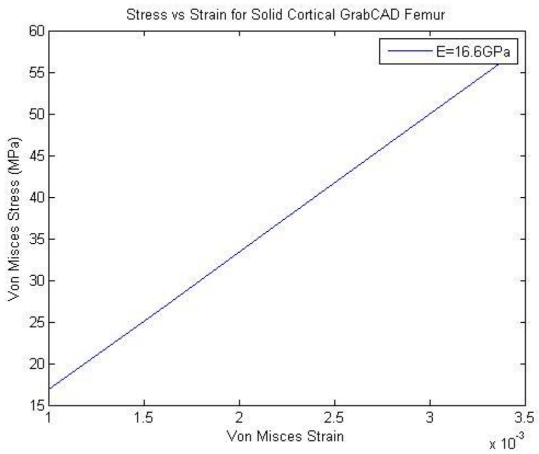

Figure 2: Stress vs Strain Plot for Solid GrabCAD Femur

In order to avoid the issues present in the Sawbones CAD file, a second 3D femur geometry was sourced from the free 3D printing website GrabCAD (Fig. 4). When compared to the Sawbones geometry, the GrabCAD femur was both very close geometrically and appeared to lack all of the difficulties present within the Sawbones file. In order to guarantee the validity of the model, the femur was given mechanical properties like a Young's Modulus of 16.6 GPa and tested to ensure that these mechanical values were returned through numerical data. The Von Misces stress and strain were taken for a 200 to 600 N load. Fig. 2 demonstrates that no fundamental flaws exist in the GrabCAD geometry, as the inputted data was returned accurately as both isotropic and linear with a modulus of elasticity of 16.6 GPa. Being no longer constrained by flexibility issues present in the Sawbones model, the GrabCAD geometry was altered in ANSYS DesignModeller Workbench with channels which mimic the location of cancellous bone (Fig. 5).

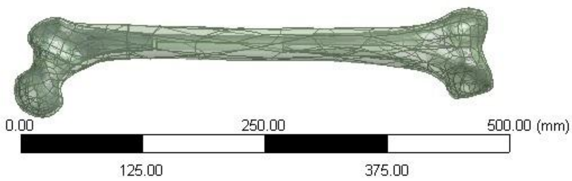

Figure 3 – Sawbones Femur Geometry

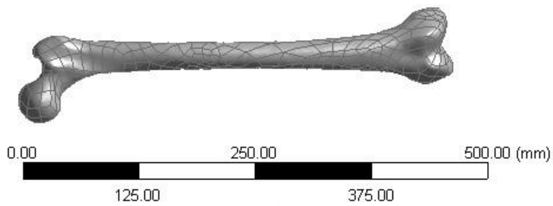

Figure 4 - GrabCAD Femur Geometry

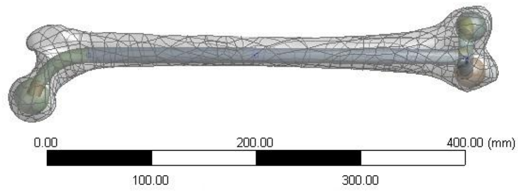

Figure 5 – GrabCAD Femur with Cancellous Channels

III. Static Finite Element Testing

C. Mesh Analysis

Given the complex curvatures inherent in organic geometries like bones, meshing can be a difficult factor in FEM. The mesh needs to be of a high enough quality to satisfy accuracy requirements, while still solving within a realistic time frame of no longer than four hours. A simple way to judge the quality of a mesh is by comparing the skewness and orthogonal quality [6]. The skewness is a measure of the distortion of an element compared to the ideal shape, while the orthogonal quality is the individual cell quality. The skewness ranges from 0 as an ideal score to 1 being unacceptable. Orthogonal quality is the opposite.

It was found that using the Automatic Meshing function with specified Face-Sizing within ANSYS provided the best mesh with the least initial penetration gap for contact pairs, while methods such as Sweep or Hex Dominant entire failed to mesh due to non-uniform geometry. Tetrahedron-based, patch independent methods would mesh, however the element size required meant the time to analyse a simple scenario became enormous. Using Automatic Meshing, the element size was altered from 2 to 7 mm and skewness and orthogonal quality values were taken. One mm measurements were also attempted, however due to a lack of computing power no solution could be found.

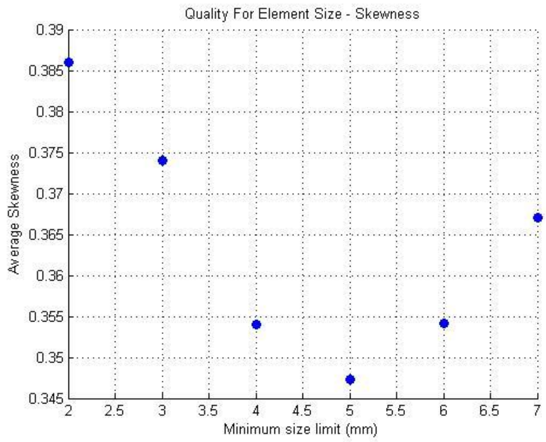

Figure 6 – Skewness for 2-7 mm element size

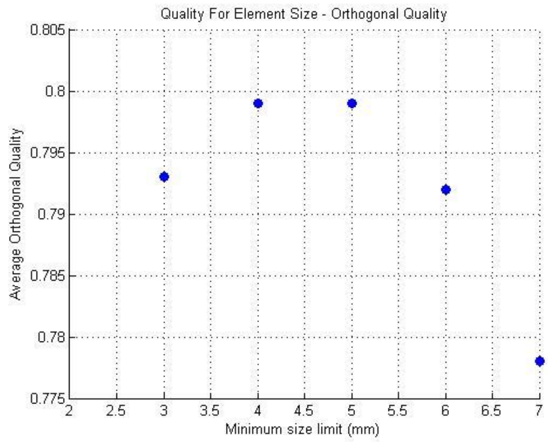

Figure 7 – Orthogonal Quality for 2-7 mm element size

Fig. 6 and Fig. 7 demonstrate that a 5 mm element size minimized the combined skewness and orthogonal quality. The physical meaning of this is that using a variety of appropriate element shapes of maximum length 5 mm, an organic, curved geometry could be meshed with the least distortion and the highest numerical quality of the individual elements.

D. Methodology

SBLT Holmes statically tested a composite femur in 2013 using a 22 kN Moog Dynamic Load Frame. The composite thigh bone was periodically subjected to an increasing compressive load up to a maximum force of 2 kN. Regression lines were fitted to the collected data, producing the graph shown in Fig. 8. Additionally, Holmes conducted a Young's Modulus test of the entirely cortical shaft, finding the fibre-reinforced epoxy to have a modulus of elasticity of 11.8 GPa. Due to an unfortunate fault with the Moog Load Frame, the composite femur was tested to failure; the force on the composite bone being no greater than 4 kN. Fig. 9 demonstrates the

placement of the composite femur in the Dynamic Load Frame, following information gained from the Bergmann, *et al* study in 2001 [2]. This level of load and angle is indicative of an *in vivo* static standing test [9].

In order to successfully validate the finite element femur model, a simplification process occurred for the structural loading which mimicked the methodology of SBLT Holmes' physical testing. Namely, the femoral head was placed under a compressive load at 27°, while the condyl section faces are constrained against displacement in the x, y and z direction. Although viewed as considerably important in dynamic finite element analysis, loading due to muscular torsion or tension was ignored in the static testing in order to simulate the load placed upon the femur while standing without movement.

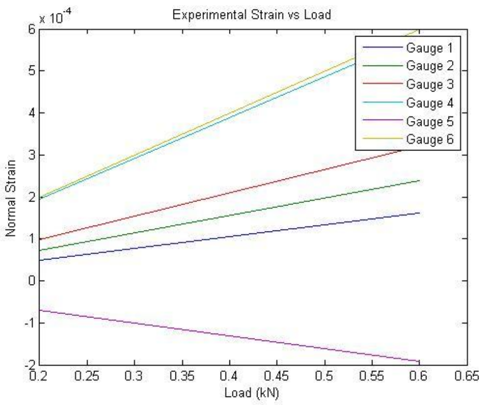

Figure 8 – Graphical representation of the regression equations fit to strain data

Figure 9 – Experimental strain test [6]

E. Mechanical Properties

In her examination of the modulus of elasticity for the cortical-equivalent material within the composite femur, SBLT Holmes found that the Young's Modulus was smaller than that quoted by the supplier, Sawbones. Given the ambiguity in having two reputable sources claim different moduli, a basic mechanical properties analysis was conducted. The Young's Modulus testing was conducted in two steps. The first was a hollow femur with only cortical bone properties and the second was the analyzed cortical bone model with cancellous properties instead of hollow channels.

The femur was loaded with 200 N and constrained around the condyl section so that no displacement could occur. The normal strain was taken for each point along the femur which matched the location of an experimental strain gauge. The slope of the numerical strain data was subtracted from the slope of the experimental strain data, then the percentage difference compared to the experimental strain data was taken.

$$\text{Percentage Difference} = \frac{\left(\frac{\text{P}}{\delta}\right)_{\text{exp}} - \left(\frac{\text{P}}{\delta}\right)_{\text{num}}}{\left(\frac{\text{P}}{\delta}\right)_{\text{exp}}} \times C \qquad \text{Eq. (1)}$$

Where C is a correction factor for using a different femural geometry as compared to the experimental model. The factor is based upon the dissimilarities within the shaft radii, due to the difficulty in calculating any other of the organic geometric parameters.

$$C = \frac{\pi r^2{}_{\text{num}}}{\pi r^2{}_{exp}} = 1.1479 \qquad \text{Eq.(2)}$$

This was initially done for a Young's Moduli of 6, 11 and 15 with data from Strain Gauge 2 as an indicator of the most appropriate range to be examined. From Fig. 11, it is apparent that the initial cortical Young's Modulus of 11 GPa had the smallest difference to the experimental data. Upon considering the data from each strain gauge, 11 GPa still afforded

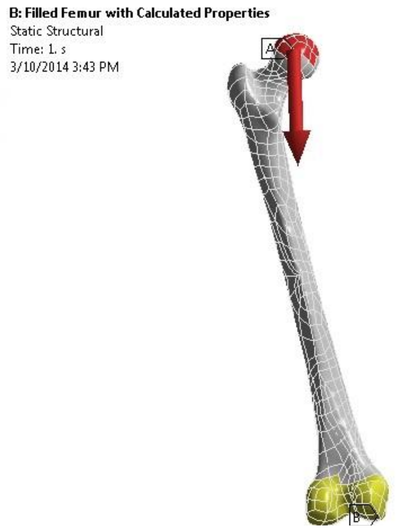

Figure 10 – FEM of the femur, which mimics experimental testing

the smallest difference to the experimental data. This agrees well with SBLT Holmes' results. Bell, *et al* concluded the following after mechanical property tests on composite Sawbones femurs; '...the modulus of elasticity of SGFR epoxy used in third generation *Sawbones* is highly temperature dependant. A reduction in the modulus of elasticity of up to 63 percent was observed when increasing the temperature of the specimens from room to body temperature.' [1] While the femur used for experiments in 2013 was a fourth generation composite bone, variation in the mechanical properties clearly exist. This could be due to temperature, the inherent manufacturing difficulty in creating a composite with a uniform distribution of reinforcing fibres or due to the incorrect assumption that the geometry of a composite test piece does not affect the stress distribution.

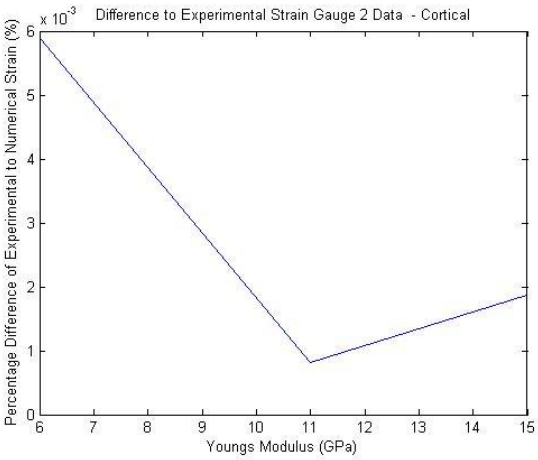

Figure 11 – Initial cortical Young's Modulus examination for 1 strain gauge

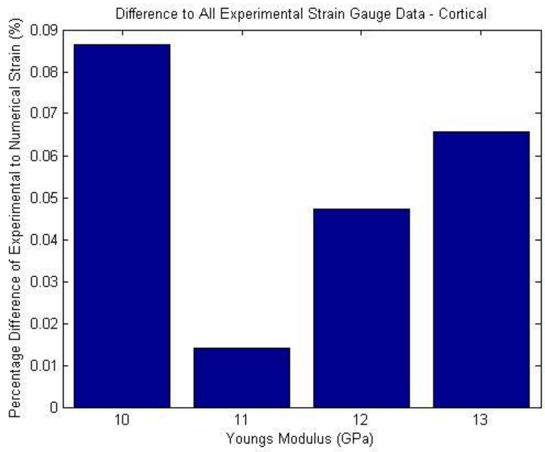

Figure 12 – Detailed cortical Young's Modulus examination for all strain gauges

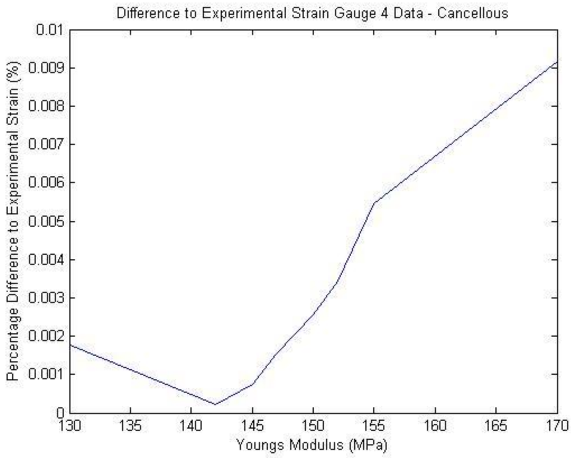

Figure 13 – Initial cancellous Young's Modulus xamination for 1 strain gauge

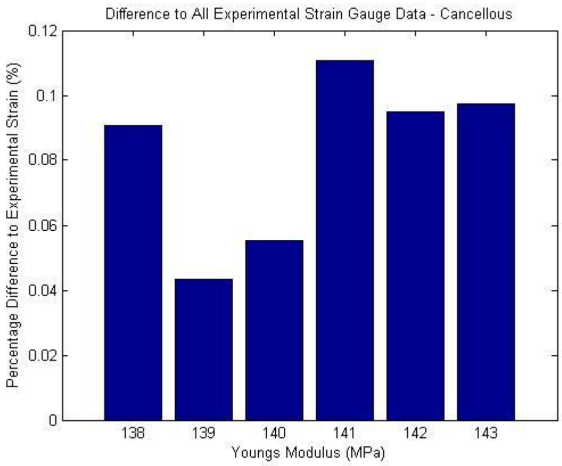

Figure 14 – Detailed cancellous Young's Modulus examination for all strain gauges

The verification of mechanical properties was repeated with the same methodology for cancellous bone. As SBLT Holmes did not conduct trabecular bone modulus of elasticity tests, a much wider range of finite element analysis was required in order to guarantee accuracy. It was found that, like the cortical bone, Sawbones had over-quoted the material properties of the cancellous-equivalent rigid polyurethane foam.

As can be seen from the Fig. 12 and 14, using a femoral geometry with both cortical and cancellous properties gives a result that has a gradient difference of 0.0433%, as compared to the hollow bone with cortical properties only, which yields a 0.0142% gradient difference. This surprising result is in fact accurate when considering the nature of the composite femur to be modelled.

Table 1 - Mechanical Property Comparison

Young's Modulus	Cortical (GPa)	Cancellous (MPa)
Sawbones	16.7	155
SBLT Holmes' Experimental	11.8	/
Numerical	11	139

Fig. 15 illustrates this point; the shaft of the femur is hollow, with the majority of the cancellous-styled material concentrated in the head and condyl sections of the femur. The shaft carries the majority of the load with the weakest point occurring in the neck of the femur where the comparatively very low modulus of elasticity; 70 times smaller, can be approximated to hollowness. Thus, it is a reasonable conclusion that the hollow femur with just cortical properties is the best representation of the composite bone. Table 1 shows the experimentally and numerically derived moduli of elasticity and well as the Sawbones quoted values for cortical and cancellous bone. All further analysis was conducted using a cortical modulus of 11 GPa and hollow inner channels.

Figure 15 - Photo of the hollow shaft interior of the experimental composite femur

F. Strength of a Femur

In order to further confirm the validity of the model with the available test data, the force required to reach the ultimate strength and create fracture in the femoral neck was examined. SBLT Holmes found that structural failure occurred in the composite femur at or before 4 kN. As this occurred during machine malfunction while the data acquisition system was offline, the rough ultimate force figure was not conducted under quasi-static conditions and instead represents a dynamic ultimate force.

Thus, this project will combat both the uncertainty in the composite mechanical properties and the uncertainty in the ultimate force value by using a minimum to maximum range of femoral strengths. In order to qualify as a valid model, the finite element femur must return a maximum stress value at the neck to be physically realistic, whilst also reaching a 4 kN force value.

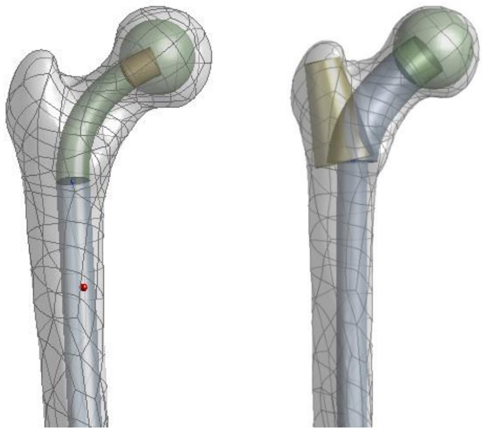

Figure 16 – a) The initial virtual femur geometry b) The updated virtual femur geometry

The first calculation for the femoral ultimate strengths utilized the uncertainty in the experimental composite femur properties. Sawbones quotes the compressive strength of their cortical-style material at 157 MPa, however given the difference between their quoted modulus of elasticity and the experimental and numerical results, this error can be used to create an initial maximum and minimum range of strength. For this, the percentage difference from the quoted Young's Modulus to the numerically calculated value was used. It should also be noted that only the compressive ultimate strength value was available from Sawbones and so the compressive and tensile strengths were assumed to be equal. This is unlikely as fibre-reinforced composites generally have higher tensile ultimate strengths.

$$\frac{16.7\text{-}11}{16.7}\times 100 = 34\% \qquad \text{Eq. (3)}$$

Thus,

$$\sigma_{uc} = (157\times 1.34) \text{ to } (157\text{-}(157\times 0.34)) \qquad \text{Eq.(4)}$$
$$= 210 \text{ MPa to } 103.6 \text{ MPa}$$

It is unsound practise to simply use a statistical range without first verifying its accuracy. Reilly and Burstein [11] found the maximum and minimum cadaveric ultimate compressive femur strength values to be 203 to 132 MPa. Koch in 1911 found the range of compressive strengths to be 126 to 165 MPa, [7]. Given a roughly 20% difference exists between the statistically calculated lower femoral ultimate strength and the value found experimentally by Reilly and Burstein and Koch, it is more appropriate to use the higher ranged minimum value. This project will use the range of strengths found by Reilly and Burstein because, being more modern, their experimentally tested cadaveric bones are more likely to have grown and lived in conditions similar to those found in the twenty-first century. The effects of diet, disease and a lack of modern medical treatment which

could have artificially lowered the femoral maximum compressive strength found by Koch, are less likely to have affected the more modern 1975 research.

The ultimate strength FEM was conducted in the same manner shown in Fig. 10, with the force placed on the femur head and the condyl sections constrained. Fig. 17 shows that a load of 4 kN exceeded the lower boundary of femoral ultimate strength. The location of maximum stress, however, occurs at an experimentally incorrect location. The location of fracture is designated by the maximum tensile stress, located at the uppermost neck of the femur, while numerically the maximum occurred in the shaft. As stated in *Mechanical Properties*, this project was initially conducted with incorrect assumptions about the internal geometry of the composite femur. Fig. 16 (a) demonstrates the internal channels initially added to the GrabCAD 3D geometry, which produced the results of Fig. 17. In order to achieve the correct maximum stress location, this project hypothesized that an alteration of the femoral neck geometry was required. This was achieved through the use of DesignModeller in ANSYS Workbench, as shown by Fig. 16 (b). The second internal femur geometry is significantly closer to the actual internal composite geometry.

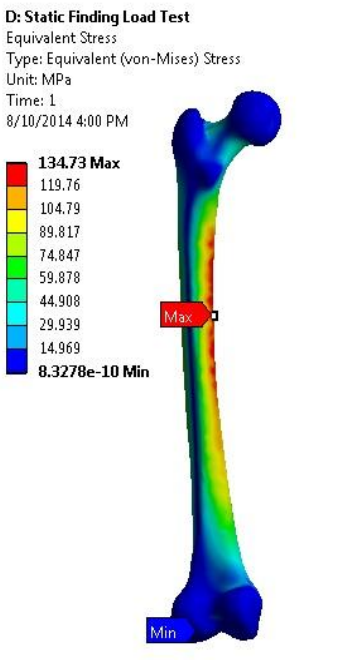

Figure 17 – 134 MPa occurred at 4000 N

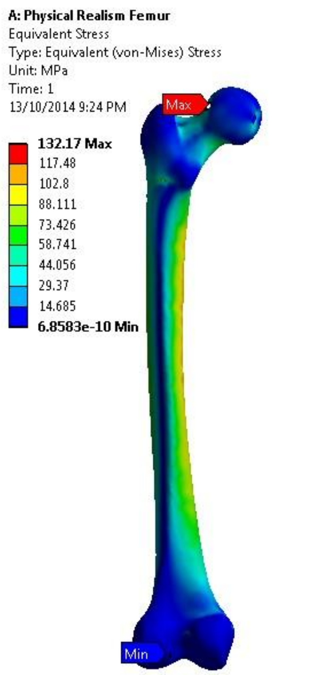

Figure 18 – 132 MPa occurred at 3390 N

The effect of this alteration is shown in Fig. 18, where a stress of 132 MPa occurred in the femoral neck, with a load below 4 kN. The maximum compressive strength of 203 MPa was achieved for a load of 5.2 kN, with the largest tensile stress also in the femoral neck. Upon decreasing the amount of cortical area in the femoral head, the finite element femur model has met both conditions stated as necessary for model verification.

IV. Dynamic Finite Element Testing

G. Dynamic Loading

Due to time constraints, conducting dynamic finite element analysis was not possible. Instead, quasi-static analysis was conducted with the loads associated with dynamic movement. Using the results of an in-vivo study conducted by Bergmann, *et al* in 2001 [2], the factor of safety was found for normal walking and walking up stairs with numerical results and ultimate strength values found by Reilly and Burstein. The factor of safety can be calculated using Eq. (5), with a result less than 1 indicating yield and fracture. An error margin of 5% exists in these results as the initial dynamic loadings came from the graphs in Fig. 19, without individual data nodes. *Annex 1* describes each force and moment taken from the Fig. 19. As the Begrmann, *et al* graphs are in terms of body weight, the weight used was the 2012 Australian Bureau of Statistic Australian male average weight of 86 kg.

$$\mathrm{FS} = \frac{\sigma_{\mathrm{uc\ exp}}}{\sigma_{\mathrm{uc\ num}} \pm 5\%} \qquad \text{Eq. (5)}$$

When conducting static finite element analysis of the verified femoral model using the maximal dynamic loads from normal walking, a stress of 112.75 MPa occurred in the femoral neck. Using the maximal loads from walking up stairs, a stress of 138.66 MPa occurred. Table 2 describes the factors of safety with a 5% error margin. The highest factor of safety occurs for normal walking at 1.89 and the lowest for walking up stairs at 0.9. As the lowest factor of safety is less than 1, this would indicate that yield had been reached and that fracture would occur. Obviously this stretches the realm of imagination. For the normal activity of walking up stairs, the average human being does not break their femur. This result stems from the inaccuracy of not considering soft tissue forces in the analysis.

Within the human body, the dynamic loads placed upon a femur are more than the resultant load on the femoral head from the body weight. While conducting dynamic tests, large discrepancies can occur when not

considering the torsional effects due to muscle contraction and expansion or the load altering effects of changes to the body weight angle because of movement. Duda, *et al* found in a FEA study on dynamic femoral strain rates, that not considering muscle activity produced a difference of 33% to the experimental strain rate. The 'anatomy and function of the mechanical system consisting of femur, pelvis and acting muscles is optimised in a way that limits the highest bending stresses in the femoral shaft' [4]. In other words, muscles acts to protect bone from excessive bending moments by applying tensile [20]. While gender, age, general health and race have a considerable effect on the mechanical properties and to a certain extent, the geometry of a human femur, that effect can only be qualitative in this project. Thus, this project concludes that by not considering the helpful effects of muscular activity, the incorrect result of a factor of safety less than 1 was achieved.

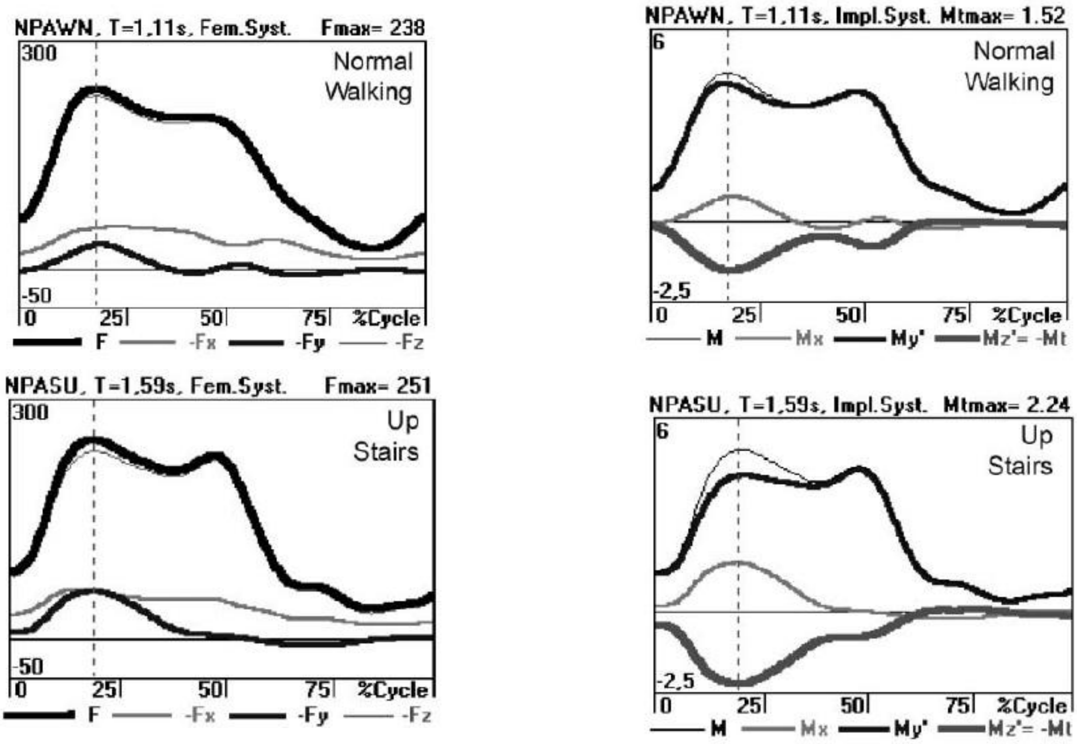

Figure 19 - from left to right a) Forces under normal walking conditions b) Moments under normal walking conditions c) Forces whilst walking up stairs d) Moments whilst walking up stairs [2]

Table 2 - Dynamic Load Factors of Safety

Numerical Stress (MPa)	Factor of Safety (high)	Factor of Safety (low)
Normal Walking: 112.75 $\pm$ 5%	1.89 - 1.70	1.23 - 1.11
Walking Up Stairs: 138.66 $\pm$ 5%	1.54 - 1.39	1.00 - 0.90

V. Recommendations

This project has resulted in the construction of a static finite element femoral model, which was verified with experimental data gained from SBLT Holmes' project in 2013. There are three areas within the project, however which require more analysis in order to improve the model's accuracy and add to the knowledge surrounding finite element femoral models.

The first recommendation is a full experimental investigation of the mechanical properties, compressive and tensile ultimate strengths of the fibre-reinforced epoxy and rigid polyurethane foam used by Sawbones to mimic cortical and cancellous material. This study should be conducted in differing temperatures and with samples from multiple composite femurs in order to understand the extent to which manufacturing processes interfere with mechanical properties. The results of this study could be used to further improve the accuracy of the FEM.

The second recommendation is that the finite element femur project be continued with a focus on dynamic analysis. An investigation into the role of musculature, ligament and tendon loads would prove informative and advance the static model. Currently, the use of maximal dynamic loads in static FEA provides results that are inconsistent with real life. The addition of soft tissue loads would likely bring the model into line with qualitatively realistic results.

The third and final recommendation is an exploration into the quantitative effects of gender, age and health on femoral mechanical properties. While this study could not be conducted using finite element analysis and would likely have to be in collaboration with a larger organization, it would vastly improve the knowledge surrounding femoral properties. The discovery of a trend in mechanical properties for any of the specific variables mentioned above would improve any FEM. Further, it would give medical practitioners an opportunity to understand the needs of their patients before they present with issues, improving the chances of correct care.

VI. Conclusions

The investigation carried out in 2014 was the second iteration of a FEM project that began in 2013 with experimental testing and FEA conducted by SBLT Rebecca Holmes. The University of New South Wales at Canberra, in collaboration with The Canberra Hospital facilitated the construction of a femoral FEM in order to positively affect orthopaedic reconstruction solutions after femoral fracture. Using commerical ANSYS FEA software and a free femur geometry available from the 3D printing website, GrabCAD, the aim of the project; to create a valid and realistic FEM of the human femur which successfully represented the effect of static and dynamic loading from both stationary and active motions, was achieved. This processes involved altering the femur geometry and analysing the mechanical properties of cortical and cancellous bone, as well as the application of loads and bending moments in static and dynamic conditions. It was found that a composite femur should be modelled as hollow with only cortical mechanical properties. The modulus of elasticity of the fibre-reinforced epoxy was found to 11 GPa, which agreed well with the value found by SBLT Holmes. Further, the validity of the femoral model was confirmed by examining a maximum to minimum range of ultimate compressive strengths. Upon decreasing the amount of cortical area in the femoral head, the model obtained its highest stress at the neck; the experimentally appropriate location, and ranged over 4 kN; the estimated force value at fracture. Finally, factors of safety were examined for maximal dynamic loads in static conditions and it was found that not considering soft tissue loads in conjunction with body weight loads led to results inconsistent with life.

Acknowledgements

I would like to thank the supervisor of the finite element femur project, Dr. Krishna Shankar for his advice and guidance. I would also like to express thanks to Hongxu Wang, Karthik Ram Ramakrishnan, Samyukta More, Cameron Sheridan and Ben Wood for their technical knowledge and assistance. Finally to my partner, Angus McLaren, thank you for your patience and the endless cups of tea over the last stressful year.

Bibliography

1. Bell, C. Weinrauch, P. Crawford, R. Pearcy, M. *Thermo-mechanical investigation of the short glass fibre reinforced epoxy used as the cortical bone analogue in sawbones femurs*, Queensland University of Technology, School of Engineering Systems, 2005
2. Bergmann, G, Deuretzbacher, G, Heller, M, Graichen, F, Rohlmann, A, Strauss, J, Duda, G. *Hip Contact Forces and Gait Patterns from Routine Activities*, Journal of Biomechanics, Vol. 34, pp. 859-871, 2001
3. *Composite Bones: Fourth Generation Femur*, Pacific Research Laboratories, 2013, viewed: 05 May 2014, available: << http://www.sawbones.com/Catalog/Biomechanical/Composite%20Bones/3403>>
4. Cristofolini, L, Viceconti, M, Cappello, A, Tonit, A. *Mechanical Validation of Whole Bone Composite Femur Models*, Journal of Biomechanics, Vol. 29, pp. 525-535, 1996
5. Hambli, R, Lespessailles, E, Claude-Laurent, B. *Integrated remodelling-to-fracture finite element model of human proximal femur behavior,* Journal of the Mechanical Behavior of Biomedical Materials, Vol. 17, pg. 89 – 106, 2012.
6. Holmes, R. *Finite Element Model of A Human Femur*, University of New South Wales, Canberra, 2013
7. Koch, J. *The Laws of Bone Architecture*, The American Journal of Anatomy, Vol. 21, No. 2, pg. 177-298, 1917.
8. Lau, C. *Anthropometric Study of the Femur – An Automated Approach*, Surgical & Orthopaedic Research Laboratories of The University of New South Wales, 2009.
9. Lu, T, Taylor, S, O'Connor, J, Walker, P. *Influence of Muscle Activity on the Forces in the Femur: An In Vivo Study*, Journal of Biomechanics, Vol 30, pp 1101-1106, 1997
10. Martens, M, Audekercke, V, Delport, P, Meester, P, Mulier, J. *The Mechanical Characteristics of Cancellous Bone at the Upper Femoral Region*, Journal of Biomechanics, Vol. 16, pp 971-983, 1983
11. Reilly, D, Burstein, A. *The Elastic and Ultimate Properties of Compact Bone Tissue,* Journal of Biomechanics, Vol 8, pp. 393-405, 1975
12. Sawbones Worldwide, Biomechanical Test Materials Catalogue, Vashon, WA
13. "distal." Merriam-Webster.com, 2014, viewed: 05 May 2014, available: <<http://www.merriam-webster.com/>>
14. "proximal." Merriam-Webster.com, 2014, viewed: 05 May 2014, available: <<http://www.merriam-webster.com/>>

Annex 1

Table 3 - Dynamic maximal force and moments used in static FEA

Normal Walking		
Forces	**Percentage of Body Weight**	**Load (N or Nm)**
Fx	230	1940.4
Fy	60	506.19
Fz	25	210.9
Mx	-2	-16.8
My	4.5	37.9
Mz	1.5	12.65
Walking Up Stairs		
Forces	**Percentage of Body Weight**	**Load (N or Nm)**
Fx	230	1940.4
Fy	75	632.7
Fz	75	632.7
Mx	-2.5	-21.11
My	5	42.2
Mz	2	16.87

Note: Body weight used was Australian male statistical average in 2012 ~ 86 kg