Analysis of Cortical Bone Fracture in Human Thigh Segment Through Finite Element

M. K. Ali Hassan, F. M. Zarzour and M. S. Salim
School of Mechatronic Engineering, University Malaysia Perlis, 02600 Arau Perlis Malaysia.
khairulhassan@unimap.edu.my

***Abstract*—Stress intensity factor, *K*, and von mises stress have been obtained by using three-point bending test in two-dimensional (2D) model of cortical bone in human thigh segment. This test has been done in finite element analysis (*K*) with respect to the continuum mechanic approach. The K values obtained from the Ansys simulation is almost similar to the K values of the equation. The differences values between them were in between 0.02% to 6.64%. The values obtained in von mises stress increases whenever the crack length decreases, while the maximum value obtained when the crack length is 1mm. The values of K is directly proportional to the ratio of crack length, where whenever the ratio of the crack length over width increases, the K values also increases.**

***Index Terms*—Cortical Bone; Finite Element; Fracture; Human Thigh; Stress Intensity Factor.**

I. Introduction

A. Finite Element Method

Finite element method (FEM) is a technique of solution of the boundary value problems. It can be explained as a numerical method for solving differential and integral equations. Finite element analysis (FEA) is the practical application of FEM. FEA is a computational tool for carrying out engineering analysis. It can be used for analysis of new product designs as well as for the existing designs using the equations of mechanics of materials [1-2]. In this work, FEA is used to analyses the human femur bone.

B. Femur Bone

Femur bone is also known as thigh bone. The thigh bone is the longest and, by most measures the strongest and heaviest bone in the human body. The length of this bone is almost 26.74% of the height of the person. The thigh bone is divided into three parts: upper extremity, body, and lower extremity. The upper part consists of head, neck and the two trochanters (the small second segment between the coxa and the bone). The body is the long and almost cylindrical in shape. It is slightly arched. The lower extremity is bigger than upper extremity. The human body is shown in Figure 1, where femur bones are marked. The bone material has been studied in depth by many workers for e.g. [3-4].

Figure 1: Femur bones are marked

C. Three-Point Bending Test

In majority cases, the three-point bending test is applied. This experiment is implemented by applying two static points or standers from the bottom and one dynamic force from the mid-top of the bone as shown in Figure 2, standers are held in measured the angels to the axis of the bone at 90º. The fracture of the bone takes place when the force applied on it increases gradually to a certain point [5].

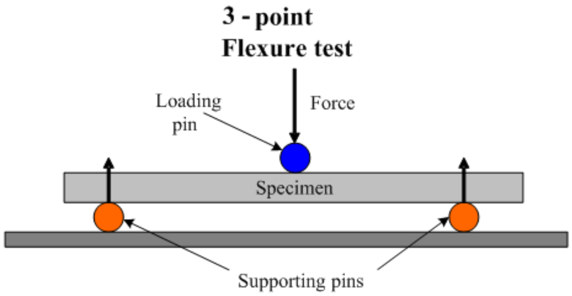

Figure 2: Schematic diagram of a three-point bending test

Kotiveerchari et al. [6] used the three-point bending test to measure the fracture of femur bone such as those which happen in car accidents; the permissible force was 115.6 N in bending test. The elastic modulus of the thigh bone was 1.68 Gpa.

Where Khor et al. [7] studied to apply a finite elements model to define the fracture of cortical bone with a femur model using asymmetric material properties in three-points bending test load case, at this study bending test performed on the tibia gave a failure load of 3549 N.

D. Bone Toughness

Numerous studies that highlighted age-related issues in the mechanical properties of bone have shown significant deterioration in bone toughness with age [8-9]. Koester et al.

[9] studied about fracture test to estimate the impact of aging in fracture toughness in transverse using R-curve, in order to distinguish the fracture mechanism of tibia loaded in 3 points bending test. Therefore age is related to the deterioration that contains quality and quantity of the bone mineral density (BMD). BMD is a responsible for the increase of elderly bone fracture. The another author also used R curve to investigate the effects of the age-related changes for the crack and growth toughness in mouse bone based to stress intensity factor measurement [10].

E. Stress Intensity Factor

To determine fracture toughness on a human cortical bone, there have been many equations used to find the stress intensity factors (SIF), and the magnitude of *K* depends on material such as the young's modulus and the Poisson's ratio in plan strain. However, there are two known formulas that are assured of determining the K factor of the 3 points bending test as stated in Equation (1) [8-11].

$$K_1 = \sigma\sqrt{\pi a}F\left(\frac{a}{b}\right) \tag{1}$$

where:

$$\sigma = \left(\frac{6M}{b^2}\right) \tag{2}$$

$$M = \left(\frac{Ps}{4}\right) \tag{3}$$

$$F\left(\frac{a}{b}\right) = \frac{1}{\sqrt{n}}\left(\frac{1.99 - \frac{a}{b}\left(1 - \frac{a}{b}\right)\left(2.15 - 3.93\left(\frac{a}{b}\right) + 2.7\left(\frac{a}{b}\right)^2\right)}{\left(1 + 2\frac{a}{b}\right)\left(1 - \frac{a}{b}\right)^{3/2}}\right) \tag{4}$$

where: σ = Applied stress
M = Bending moment
B = Depth of bend specimen
P = Distributed load
s = Span between supports of bend specimen
a = Depth of edge crack
$F\left(\frac{a}{b}\right)$ = Configuration correction factors for stress intensity factor

Numerical studies also highlighted the importance of understanding the fracture mechanism and behavior of bone, since the bone is subjected in a multiple loading. Moreover, they used finite element method on three-dimensional (3D) model of cortical bone of human tibia and thigh, and by using many types of loading test on different bone's long such as three or four-point bending test and cantilever test to measure the stress intensity factor *K*.

In this study, a two-dimensional (2D) model of cortical bone for human thigh segment will be developed by using three-point bending test. The stress intensity factor, *K* will be evaluated from the model. Hence, mapping contour of stress intensity of the model will also be discussed.

II. Methodology

The flowchart shown in Figure 3 describes the strategy that has been applied to achieve the objective of this project. In order to implement cortical bone fracture, a study of how to use Ansys mechanical APDL software. Moreover, a study of developed two dimension of human thigh segment, determine the fracture behavior of cortical bone segment for three-point bending test, to identify the stress intensity factor (K) at different length of crack and applied loads, analyze the accuracy of the K values obtained from simulation by comparing with the theoretical values obtained based on formula or other previous studies, this will discuss in the following sections.

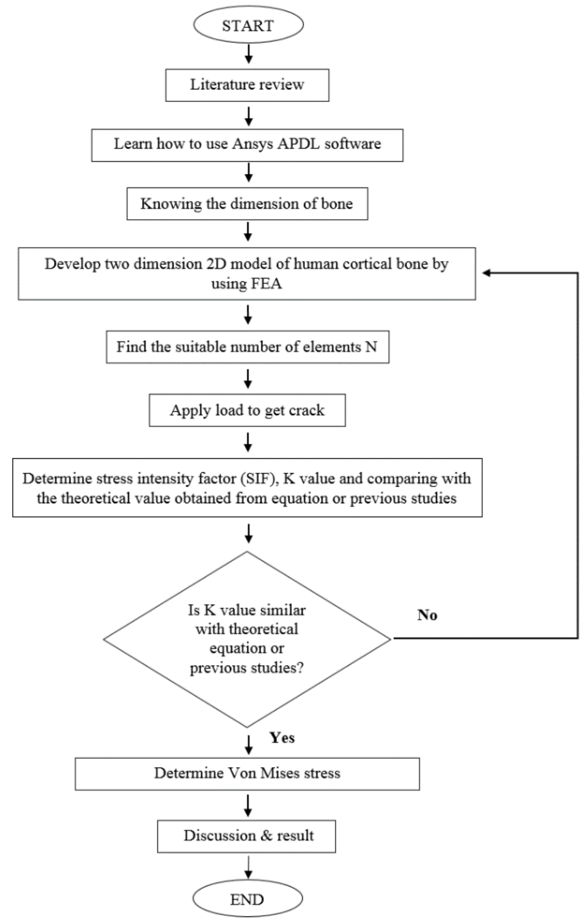

Figure 3: Project flowchart

A. Model of Bone

Finite element models of bone will develop by Ansys APDL software, based on dimensions of bone adapted from human measurement where the parameter of the femur bone is used in the program to develop a two-dimensional model. Where from this model it can be stated that the force applied to the femur bone can be approximately determined, in other word the femur bone breakpoint can be measured using this 2D model.

B. Find the Accurate Number of Element

The number of elements is an important item to create a manual meshing, and finding an accurate number that can help in identifying the close K-ansys value of K-equation value for the dependent value in the project result. Therefore to find it there should be few steps that should be done at first after the shape is done. After developing the shape, there should be section created in concentrate KPs. To be done, the user needs to go through preprocessor section and choose meshing alongside. After that, the user needs to fill up the DELR radius of 1st row of elements. The value of DELR will be obtain by dividing the crack value by the number of elements. The NTHET is the number of an element chosen previously. The NTHET value could be any number between 4 and 10 as long as the Kansys is close enough to the value of K-equation .the KCTIP value is at 1⁄4 pt.

C. Determination of Stress Intensity Factor (SIF), K

Stress intensity factor measurement is based on the dimensions of bone, the bone toughness and bending test that will choose in this project. The fracture will be modelled in three-points bending test, by applying one dynamic force from the mid-top of the shape of a cortical bone and apply several loads until the crack happens on the applied force point, based on the bone shape dimensions. From there, the K value will be obtained, and it will differ depending on the factors mentioned before.

After obtaining the value from the model, there is a different method to obtain the value of K which is by using theoretical equation. In this way, there will be a comparison between the value of K obtained from the model and the value of K obtained from the equation. The result of K values should be somewhat comparable, this way the build-up design will give us a closer result of fracture intensity of femur bone compare to a real injury obtained from the previous paper.

III. Testing and Analysis

A. Experimental Result

The deformed shape of the model after all the loads being applied shown in Figure 4. The crack growth was found on the midline of the model parallel to the applied load, P = 1000 N. There is no other deform area on the model because the load is applied at the center of the model and other two constraints are distributed equally on the left and right-hand side.

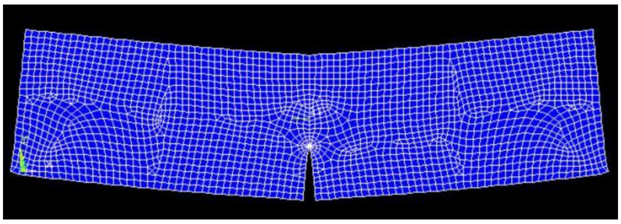

Figure 4: Deformation of cortical bone model for crack length, a = 5 mm

From Table 1, it is concluded that the SIF directly proportional to the ratio of crack length and width of the model. As we can see the values of SIF and the values using ansys module, we can conclude that the crack length of the cortical bone will the values of K_1. Comparing the differences in values between $K_{equation}$ and K_{ansys}, it can be concluded the difference error is between (0.0216 to 3.68). the range of SIF (k) in value differ where the minimum value is 25653.42 and the maximum value is 250643.9 in term of the equation. However the minimum value of ansys is 24708 and the maximum value is 256200.

Table 1
Summary of Stress Intensity Factor, K with Respect to Different Crack Length (a) with 1000N Applied Load

w (mm)	a (mm)	a/w	$K_{equation}$ ($Nm^{-3/2}$)	$K_{1\ ansys}$ ($Nm^{-3/2}$)	$K_{2\ ansys}$ ($Nm^{-3/2}$)	Error (%)
13.5	1.0	0.074	25653.42	24708	0.00345	3.68
	2.0	0.148	34900.37	34136	0.73297	2.19
	3.0	0.222	43047.20	42502	0.48335	1.26
	4.0	0.296	52056.9	51609	2.9737	0.86
	5.0	0.37	63121.43	62655	3.3789	0.738
	6.0	0.44	77046.7	77030	2.5028	0.0216
	7.0	0.52	97905.8	96875	3.4342	1.05
	8.0	0.59	126447.5	126080	7.7535	0.29
	9.0	0.67	172447.99	172740	5.8983	0.16
	10.0	0.74	250643.9	256200	29.706	2.21

Figure 5 shows how close the difference in values between the $K_{equation}$ and K_{ansys} and we can see also the trend of the graph is similar to each other as the ratio of a/w increased.

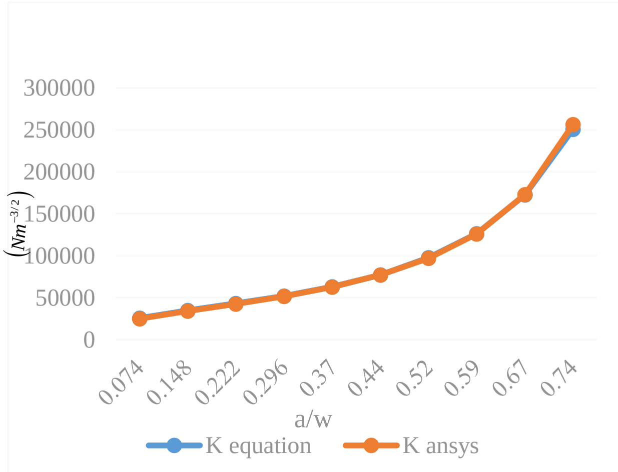

Figure 5: $K_{equation}$ and K_{ansys} versus crack length over width a/w of model at 1000N

IV. Conclusion

This study presents a method for finding a stress intensity factor, K and von mises stress by using a three-point bending test in finite element method. And all the main objectives of this research have been completed starting from the first stage until the last stage. It can also provide a better view on the design of surgical intervention tools. Thus, this paper proposed the theoretical and numerical element to investigate the fracture toughness in human cortical bone.

References

[1] Szabo, B.A. and I. Babuška, Finite Element Analysis 1991: Wiley.
[2] Zienkiewicz, O.C., R.L. Taylor, and J.Z. Zhu, The Finite Element Method: Its Basis and Fundamentals: Its Basis and Fundamentals 2005: Elsevier Science.
[3] Burstein, A.H., et al., The ultimate properties of bone tissue: The effects of yielding. Journal of Biomechanics, 1972. 5(1): p. 35–44.
[4] Katz, J.L. and Y. Hyo Sub, The Structure and Anisotropic Mechanical Properties of Bone. Biomedical Engineering, IEEE Transactions on, 1984. BME-31(12): p. 878–884.
[5] S. Seth, P. Kalra and R. Sekhar Dondapati, "Effect of Varying Inter-Implant Distance in a Two Implant-Three Prosthetic Unit Dental System: A Finite Element Analysis Study", European Modelling Symposium, pp. 291 - 296, 2014.C. J. Kaufman, Rocky Mountain Research Lab., Boulder, CO, private communication, May 1995.

[6] A. Chennakesava Reddy and B. Kotiveerchari, "Simulation of Femur Bone Fracture in Car Accident using CT Scan Data and Finite Element Analysis", *International Journal of Science and Research (IJSR)*, vol. 4, no. 11, pp. 1805-1807, 2015.
[7] F. Khor, B. Watson, M. B. Panzer, D. Cronin, S. Malcolm and D. Gierczycka, "Effect of Cortical Bone Material Asymmetry Using a Three- Point Bend Test Simulation", international Research Council on Biomechanics of Injury (IRCOBI), pp. 18-19, 2016.
[8] N. N. Mansor, R. Daud, K. S. Basaruddin and Y. Bajuri, "Review of Biomechanical Modelling of Cortical Bone Stress Fracture", 2015 2nd International Conference on Biomedical Engineering (ICoBE), 2015.
[9] R. Nalla, J. Kruzic, J. Kinney, M. Balooch, J. Ager and R. Ritchie, "Role of microstructure in the aging-related deterioration of the toughness of human cortical bone", Materials Science and Engineering: C, vol. 26, no. 8, pp. 1251-1260, 2006.
[10] A. Carriero, E. Zimmermann, S. Shefelbine and R. Ritchie, "A methodology for the investigation of toughness and crack propagation in mouse bone", Journal of the Mechanical Behavior of Biomedical Materials, vol. 39, pp. 38-47, 2014.
[11] H. Tada, P. Paris and G. Irwin, The stress analysis of cracks handbook, 3rd ed. New York: ASME Press, 2000, p. 58.