NUMERICAL SIMULATION OF STRESS AMPLIFICATION INDUCED BY CRACK INTERACTION IN HUMAN FEMUR BONE

NOOR ALIA BINTI MD ZAIN

UNIVERSITI MALAYSIA PERLIS

2014

Numerical Simulation of Stress Amplification Induced by Crack Interaction in Human Femur Bone

By

Noor Alia binti Md Zain
(1332120826)

A dissertation submitted in partial fulfillment of the requirements for the degree of
Master of Science (Engineering Mathematics)

Institute Mathematics Engineering
UNIVERSITI MALAYSIA PERLIS

2014

ACKNOWLEDGEMENT

First of all, praise to Allah who had given me the strength, courage and good health to complete this dissertation for completing MSc. Engineering Mathematics. There are several people were involved in the completion of this study, those who have provided me with invaluable information's, experimental feedback, views, comments and support. Without them, I would never have been able to finish this project.

In particular, firstly I would like to extend my appreciation to Mr Wan Zuki Azman Wan Muhamad, my supervisor and Dr Ruslizam Daud, my co-supervisor for the information, guidance, knowledge and support to begin my project until this study is completely done.

Also my acknowledgment is dedicated to my friends and team member, Siti Aisyah, Norhasyimah, Nurul Najwa and Huskhazrin whom helped me immensely by helping me a lot throughout this project. Other than that, a special gratitude to Dr Zaini as our program coordinator for the efforts to manage and guide me in the dissertation sessions. In addition, thanks to all members and staff of Institute of Mathematics Engineering program and not to forget my beloved family for all the caring, encouragement and support they given me.

Last but not least, thank you to all other contributors during the development process to complete this Dissertation study.

Simulasi Tegsan secara Amplifikasi disebabkan oleh Interaksi Retak di Tulang Paha Manusia

ABSTRAK

Kajian ini adalah mengenai simulasi berangka yang menggunakan kaedah pengiraan kajian mengenai pembesaran tegasan disebabkan oleh interaksi retak pada tulang paha manusia. Retak di tulang paha manusia biasanya berlaku kerana beban besar atau tekanan dikenakan ke atasnya. Biasanya, retak tersebut mengambil masa yang lama untuk pulih sepenuhnya. Interaksi pada retak itu masih tidak dapat difahami dengan sempurnanya kerana kerumitan pada tulang. Oleh itu, keadaan tulang yang retak itu sering kali dipandang ringan dan andaian tersebut adalah tidak tepat. Kajian ini bertujuan untuk menyiasat kesan geometri pada satah yang mempunyai dua retak kepada faktor keamatan tegasan atau "stress intensity factor" (*K*) di tulang paha. Selain daripada itu, kajian ini juga memberi tumpuan dalam kesan penguatan terhadap satah yang mempunyai dua retak di hujung model tersebut, di mana model berangka dibangunkan menggunakan kaedah pengiraan. Konsep mekanik patah dan pendekatan berangka digunakan untuk menyelesaikan masalah retak berinteraksi menggunakan teori linear mekanik patah elastik (LEFM). Maka, ia adalah penting untuk mengkaji apakah parameter yang boleh mengurangkan penyebaran retak untuk mencegah tulang tersebut daripada musnah keseluruhannya. Kajian ini telah menunjukkan bahawa had interaksi retak (CIL) dan had penyatuan retak (CUL) wujud dalam model yang dibangunkan. Untuk penambahbaikkan model ini pada masa hadapan, beberapa kaedah telah dibangunkan. Contohnya seperti menggunakan beban yang berbeza-beza, memberi ketebalan kepada model dan juga menggunakan teori atau kaedah yang berlainan dalam pengiraan faktor keamatan tegasan (*K*).

Numerical Simulation of Stress Amplification Induced by Crack Interaction in Human Femur Bone

ABSTRACT

This research is about numerical simulation using computational method which study on stress amplification induced by crack interaction in human femur bone. Crack in human femur bone usually occur because of large load or stress applied on it. Usually, the fracture takes longer time to heal itself. The crack interaction is still not well understood due to bone complexity. Thus, brittle fracture behaviour of bone may be underestimated and inaccurate. This study aims in investigating the geometrical effect of double co-planar edge cracks on stress intensity factor (K) in femur bone. Other than that, this research also focus in the amplification effect on fracture behaviour of double co-planar edge cracks, where numerical model is developed using computational method. The concept of fracture mechanics and numerical approaches to solve interacting cracks problems using linear elastic fracture mechanics (LEFM) theory is used. So, it is important to study what is the parameter that can minimize the crack propagation to prevent complete failure. This study has shown that the crack interaction limit (CIL) and crack unification limit (CUL) exist in the model developed. In future development of this research, several improvements will be made such as varying the load, applying thickness on the model and also use different theory or method in calculating the stress intensity factor (K).

TABLE OF CONTENTS

CHAPTER 2 LITERATURE REVIEW

CHAPTER 3 METHODOLOGY

CHAPTER 4 RESULTS AND DISCUSSION

LIST OF TABLE

LIST OF FIGURES

LIST OF ABBREVIATIONS

K	Stress intensity factor
G	Elastic energy release rate
J	J-integral
CTOD	Crack tip opening displacement
CUL	Crack unification limit
CIL	Crack interaction limit
SIL	Stress interaction limit
FEA	Finite element analysis
LEFM	Linear elastic fracture mechanics
K_c	Critical stress intensity factor
a	Crack length
b	Crack interval
s	Crack distance
w	Width of the model developed in ANSYS
L	Length of the model developed in ANSYS
2D	Two dimensions
GUI	Graphical user interface
DEM	Displacement extrapolation method

CHAPTER 1

INTRODUCTION

1.1 Background of study

Bone is the main component in the skeletal system in our body. Bone plays important roles especially to support our body. There are 206 bones inside an adult skeleton system (Marieb, 2009). The bone that is responsible to carry our body weight is located at the lower part of our body which is called "femur" bone. The femur bone is located in the thigh region. It is the strongest bone in the body. At the end of the bone which is in the proximal area (near to the midpoint of the body) it can be seen that a ball-like head and a neck at the structure of the bone. It extends from the knee joint and hip joint. Figure 1.1 shows the anatomy and structure of the human femur bone.

Figure 1.1: Anatomy of the human femur bone (Marieb, 2009)

The femur bone is the strongest bone in the human body. If there is fracture or crack happening within the bone, there must be large force acting on it. Fracture in a bone can cause impairment to it, where the bone cannot bear the load of the person's weight. Fracture happens when there is an initial crack occurring when certain amount of load and stress acting on the bone repeatedly. The stress at the tip of a sharp crack has the highest stress which can lead to failure on the material. This explains why small cracks can contribute to the failure of the entire structure. There are a few parameters of fracture that are widely used (Rudraraju, S. S. ,2004). These include the stress intensity factor (*K*), Elastic energy release rate (G), *J*-integral (*J*), and crack tip opening displacement (CTOD).

In this research, the main focus is to simulate stress amplification induced by crack interaction. It is based on the stress intensity factor (K) calculation using the stress singularity. This simulation is based on the changes of stress amplification in different crack intervals towards the human femur bone. Stress amplification in a solid, with the existence of crack is the amount of load that exhibits onto the solid perpendicularly. Other than that, crack unification limit (CUL), crack interaction limit (CIL) and stress interaction limit (SIL) also can be calculated.

1.2 Problem statement

Human femur bone is the most important bone inside a human body as it is responsible to support our body weight. Femur bone is the strongest bone inside our body. Hence, if fracture occurs, this means that a very large load has been applied to the bone. Usually, the most common fracture in human femur bone is caused by car accidents, falling from a height, sport activities or simply the decrease in bone mineral (osteoporosis) which usually occurs in elderly people. If fracture occurs inside the bone, it might take weeks to months or even years for a complete healing. This is why bone fracture is considered a critical problem in real life. There are several biomechanics properties of the bone that need to be understood. One of them is the stress amplification interaction within the bone. This property is important because the effect of stress amplification on the crack is known, hence estimation of healing process in bone fracture can be done. A better understanding can be done through a numerical simulation in this project. Numerical simulation is important because it provides tools that can analyze the crack problem in a given boundary condition.

1.3 Objectives

The objectives of this project are as follows:

1. To develop a numerical model for amplified interaction between double co-planar edge cracks in femur.
2. To validate and compare the numerical model with available numerical and analytical data of literature in femur bone.
3. To investigate the geometrical effect of double co-planar edge cracks on the stress intensity factor (K) in femur bone.
4. To evaluate the amplification effect (crack interaction limit, CIL) on the fracture behavior of double co-planar edge cracks in femur bone.

1.4 Scope of study

The scope of this project is to relate the mathematical modeling of human femur bone with biomechanical properties of the bone. The human femur bone was assumed as an isotropic linear elastic material so that the amount of the stress intensity factor (K) at the crack tip can be calculated using stress singularity calculations. This calculation is done by the numerical simulation available in ANSYS software. The value of K will give the idea of the stress level act upon the crack tip.

1.5 Research significance

The research significance of the project include the interaction of various parameters with the available crack. The interaction is important because in the medical sector, the interaction of the cracks indicates the human pain within the femur bone. Several implants of artificial bone have been used widely to replace fractured bone helping to recover the pain in bone. Hence, if the optimum value of the interaction which is the unification limit is known, the pain can be reduced. Other than that, this project also provides the maximum limit of crack interaction based on several parameters. This is vital because in real life situations, human femur bone will be completely damaged if the parameters exceed certain values, such as the fact that the load is exerted too much on the bone or that the crack length is too large.

1.6 Organization of the dissertation report

Chapter 1 is the introduction, which gives an outline of the research and the structure of the thesis. This chapter consists of some background of this project study, the problem statements and the objectives that must be achieved as the project is completed.

Chapter 2 discusses from the previous studies' perspectives and findings related to this project study. It introduces the structure of a human femur bone, fracture parameter of the bone, the stress intensity factor (K), the literature review of crack failure in bones and literature review of the numerical analysis in bones.

Chapter 3 explains on the project's methodology used to execute the project's analysis. One of the main stages that are important during this analysis is remodeling the design of human femur bone using the finite element analysis (FEA).

Chapter 4 discusses the results obtained by supporting several findings derived from previous studies. In this chapter, several stress parameter effects on K values have been discussed. The result of K values based on different types of parameter is constructed into a graph and will be discussed further.

Chapter 5 represents the conclusion of the projects accomplished. The advantages and future enhancement of this project were pointed out. At the end of the chapter, recommendations for future development are also provided.

.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

An adult human bone can be classified into two types which are the cortical bone (compact bone) and trabecular bone (spongy bone). Spongy bone is less dense and weaker compared to the compact bone. The function of the spongy bone is more metabolic and not much on supporting the human body. Compact bone facilitates the main function of the bone which is to support the human body and cradle its soft organs (Marieb, 2009). For example, the bone at the lower part of a human body is responsible in supporting human body trunks when standing. Other than that, the compact bone is important to protect soft body organs. The human skull provides a snug enclosure to the brain to protect the brain from any injury. Compact bone is harder and stronger compared to the spongy bone. Human femur bone is one example of compact bone as the function is to support human body weight, also allowing us to walk, run and jump.

Human femur bone consists of cortical (dense) and trabecular (cellular) tissues. This can be classified as a complex biological tissue but it can be modeled in the finite element analysis (FEA) whereby because of the macro level, it exhibits an elastic linear behavior for loads during daily activities (Huang, Z. ,2011).

2.2 Biology of human femur bone

Human bone is composed of several different types of material. Bone is considered to be a composite material which is composed of collagen, water, and mineral nano-particles. It is permeated with pores and has fluid filled within the pores. Usually the bone fracture is determined by its mineral density and the age factor. However, there is a recent research which shows that decreasing bone mineral density is not the only factor that can lead to fracture mechanism (Doblare et. al., 2004). There are several more parameters that can influence the fracture mechanism.

At the microstructure level of bone, it is clear that every particle of the bone is essential in protecting the bone from fracturing. Usually, bone has a very high elastic modulus which can endure large load applied to it but once the load exceeds the maximum load, crack and fracture can occur.

The strain rate behaviour of the bone depends on the bone microstructure. Human bone is porous while metal is considered as a non-porous element. In the previous research done by (Ani Ural et al., 2011) the simulation done by the finite element modeling shows that the amount of porosity does not affect the fracture's toughness.

Human femur bone shows off its elastic and brittle characteristics. Bone is the strongest in compression, weakest in shear, intermediate in tension. A previous research (Pithioux et. al., 2004) has shown that bovine bone is four times more brittle in dynamic load rather than the quasi-static load. This is due to the structure of the bone which consists of a fibrous structure (Pithioux et. al., 2004). The research has been done experimentally using bovine bone because the bovine bone's structure is almost similar to human bone.

2.3 Artificial human femur bone

Human femur bone is the most important bone inside the human body. Thus, if a crack inside the bone propagates and causes total failure, an artificial bone is usually implanted into the human body. Usually, a crack will grow at the neck site in femur bone caused by the high impact of stress at the site. Thus, an artificial total hip replacement will be inserted into the human body. Figure 2.1 shows the geometrical model of total artificial hip replacement.

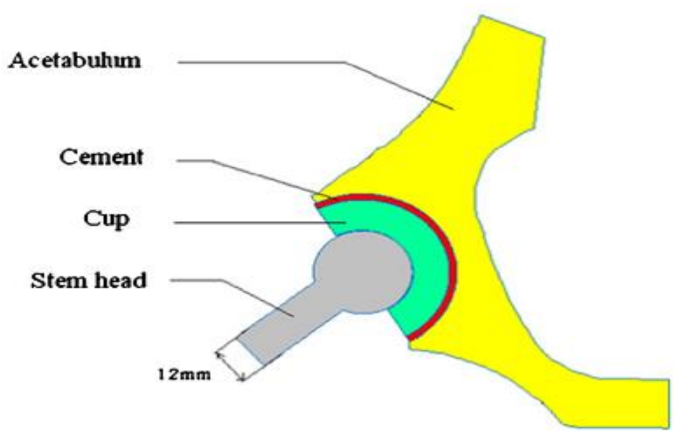

Figure 2.1: Geometrical model of total artificial hip replacement (Benbarek et al., 2013)

A previous study (Benbarek et al., 2013) has analyzed the propagation's path of the crack in the orthopedic cement of the total hip replacement and proven that the stress intensity factor and crack length increase in Mode I while decreasing in Mode II

2.4 Fracture parameter

There are three parameters which are used to characterize the stress which cause a component to fracture. These include the stress intensity factor (*K*), Elastic energy release rate (G) and *J*-Integral (*J*). The stress intensity factor (*K*) is the stress field that surrounds the crack tip of an isotropic linear elastic material. The stress can be formulated into this equation:

$$\lim_{r \to 0} \sigma = \frac{K}{\sqrt{2\pi r}} f(\theta) \tag{2.1}$$

where *K* is the stress intensity factor but this equation is only valid near the crack tip. The $\frac{1}{\sqrt{r}}$ represents the stress field around the crack. Thus the value of *K* determines the level of stress magnitude around the crack tip which has been done by previous researcher (Rudraraju, S. S. ,2004). .

The value of K should not exceed the value of the critical crack intensity factor (K_c) because if the value of K is bigger than K_c, the material will fracture (Huang, Z. ,2011).

$$K < K_c \tag{2.2}$$

Elastic energy release rate (G) is a vital parameter to be used in understanding the fracture tendency. It also refers to the energy available for crack propagation. It can be calculated using this formula:

$$G = \frac{-\partial\left(U - V\right)}{\partial A} \tag{2.3}$$

where U, V and A are potential energy for crack growth, work associated with any external forces acting and crack area.

From this formula, it shows that the value of G is the change of potential energy per unit change of area. The first law of thermodynamics dictates that if a system of non-equilibrium state changes towards the equilibrium state, the energy of the system will decrease. In this case where the system has a loaded material, the strain energy will be increased. Therefore, the net energy will also increase. Hence, the system is moving away from the equilibrium as the net energy increases. If crack propagates, there will be some energy loss from the strain energy. Such an occurrence constitutes the explanation for the fact that the energy release rate is considered as one of the important parameters.

J-Integral (J) is used to calculate the intensity of a singularity field without knowing the exact shape of the field in the vicinity of the singularity. The equation of *J*-Integral was derived from the conservation law. J is dual equivalent to K and G in the Elastic Fracture

Mechanics (EPFM). The equation of *J*-Integral done by Rudraraju, S. S. (2004) is expressed as follows:

$$J = \int_{r} w dy - \int_{r} (t_x \frac{du_x}{dx} + t_y \frac{du_y}{dy}) ds \tag{2.4}$$

where,

$Y =$ Path that surround the crack tip

$w =$ Strain energy density (strain energy per unit volume)

$t(x) =$ Traction vector along *x*-axis

$t(v) =$ Traction vector along *y*-axis

$\sigma =$ Component stress

$n =$ Unit normal vector to path *y*

$y =$ Distance along path *y*

The Linear Elastic Fracture Mechanics (LEFM) assumption suggests that every material is isotropic and linear elastic. Based on the theory of elasticity, the stress intensity factor (*K*) near the crack tip can be calculated. Crack propagation will occur if the value of *K* exceeds the material fracture toughness which is also known as the critical stress intensity factor (K_c). It depends on the mode of fracture deformation.

T-stress is also calculated at the crack tip which is important to analyze the stability analysis of the cracks in linear elastic and linear plastic behaviour of mechanics. The value