Numerical Simulation of Stress Shielding Induced by Crack Interaction in Human Phalanx Bone

by

Siti Aisyah Binti Abdul Halim
(1332120845)

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

Institute of Engineering Mathematics
UNIVERSITI MALAYSIA PERLIS

2014

ACKNOWLEDGEMENT

First of all, praise to Allah s.w.t who had given me the strength and health to complete this dissertation thus completing MSc. Engineering Mathematics. Firstly, I would like to say thank you to Mr. Wan Zuki Azman Wan Muhamad, my supervisor and Dr. Ruslizam Daud, my co-supervisor, for their knowledge and helpful guidance throughout this project. Their knowledge and guidance is very important to me to finish this project.

Also, my acknowledgment is dedicated to all of my friends for their helping throughout this project. Then, a special gratitude to my coordinator program, Dr. Muhammad Zaini Ahmad for his effort to manage and guide me in the dissertation sessions. Not forgotten to all members and staff of Institute of Engineering Mathematics and my beloved family for all the caring, encouragement and support they had given towards me.

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

TABLE OF CONTENTS

PAGE

CHAPTER 1 INTRODUCTION

CHAPTER 2 LITERATURE REVIEW

CHAPTER 3 RESEACRH METHODOLOGY

CHAPTER 4 RESULTS AND DISCUSSION

CHAPTER 5 CONCLUSION AND RECOMMENDATION

APPENDICES

LIST OF TABLE

LIST OF FIGURES

LIST OF ABBREVIATIONS

LEFM	Linear elastic fracture mechanics
SIF	Stress intensity factor
FEA	Finite element analysis
2D	Two-dimensional
CIL	Crack interaction limit
CUL	Crack unification limit
SEM	Scanning electron microscope
DCB	Double cantilever beam
3D	Three dimensional
µCT	Micro computed-tomography
K	Stress intensity factor
G	Elastic energy release rate
J	*J*-integral
CTOD	Crack tip opening displacement
SSD	Stress shielding damage
t	Thickness of model developed in ANSYS
h	Height of model developed in ANSYS
w	Width of model developed in ANSYS
a/w	Crack-to-width ratio
Cr	Crack
a	Crack length
b	Crack interval
GUI	Graphical user interface
APDL	ANSYS parametric language
DEM	Dispalcement extrapolation method
ERR	Energy release rate

Simulasi Berangka Tekanan Melindungi Teraruh oleh Interaksi Retak di Tulang Jari Manusia

ABSTRAK

Patah tulang adalah kecederaan biasa dalam kehidupan seharian. Kebanyakannya, ia meninggalkan kerosakan kekal dan memerlukan tempoh masa yang panjang untuk proses pemulihan. Keadaan ini boleh dielakkan jika kita memahami mekanik dan proses patah tulang. Kajian ini bertujuan untuk menilai tekanan pelindung oleh interaksi retak menggunakan model yang mudah berdasarkan Mekanik Linear Elastik Patah (LEFM). Simulasi yang dilakukan adalah berdasarkan penentuan Faktor Keamatan Tegasan (SIF) dan perubahan tekanan pelindung dalam keadaan retak berbeza terhadap tulang ruas manusia. Simulasi berangka telah dijalankan dalam projek ini untuk memahami tekanan pelindung yang disebabkan oleh interaksi retak. Keputusan menunjukkan bahawa interaksi dua retak adalah berkadar terus dengan magnitud SIF dan faktor interaksi pada hujung retak. Retak selari telah meningkatkan kesan pelindung apabila selang retak bertambah. Had interaksi retak (CIL) dan had penyatuan retak (CUL) juga telah diperolehi bagi setiap selang retak dalam projek ini. Beberapa penambahbaikan boleh dijalankan untuk pembangunan masa depan kajian ini, termasuklah pelbagai tekanan dikenakan kepada model, elemen berliang ditambah dalam model, satah yang berbeza pada model dan menggunakan pelbagai kaedah dalam pengiraan faktor keamatan tegasan (SIF).

Numerical Simulation of Stress Shielding Induced by Crack Interaction in Human Phalanx Bone

ABSTRACT

Bone fracture is an injury not uncommon to everyday life. Most of the time, it leaves permanent damage and a long period of recovery. This situation can be prevented if we understand the mechanics and the process of the bone fracture. This study aims is to evaluate stress shielding induced by crack interaction using a simple model based on Linear Elastic Fracture Mechanics (LEFM). This simulation based on the determination of the Stress Intensity Factor (SIF) and the changes of stress shielding in different crack interval towards the human phalanx bone. Numerical simulation had been carried out in this project to understand the stress shielding induced by crack interaction. The results revealed that the interaction of two cracks is directly proportional to the SIF magnitude and interaction factor at the crack tips. The parallel cracks have experienced increasing shielding effect as the cracks interval increase. The crack interaction limit (CIL) and crack unification limit (CUL) also had been accomplished for every range of crack interval in this project. Several improvements will be conducted for future development of this study, including various stresses loading subjected to the model, porous element added in the model, different planes of the model and use various methods in calculating the stress intensity factor (SIF).

CHAPTER 1

INTRODUCTION

1.1 Research Background

Bone fracture is an injury not uncommon to everyday life. Most of the time, it leaves permanent damage and a long period of recovery. This situation can be prevented if we understand the mechanics and the process of the bone fracture. Bone is the primary structural component of the body, serving as a protective load-bearing skeletal framework. Bone consists of heterogeneous tissue. There are two types of bone tissue; compact bone and spongy bone. These types of bone tissue differences in density and how tightly the tissue is packed together. The cells of compact bone, known as cortical bone, appear to be tightly packed into a solid mass. However, this type of bone is not completely solid as there are small canals (blood vessels) run through the bone. For spongy bone, this bone has large bone spaces and resembles a sponge. Therefore, spongy bone is lighter and less dense than compact bone (Doblare et al., 2004 and Marieb, 2009). Thus, this research study will concentrate on fracture phenomenon on cortical bone.

Fracture bone can occur in all parts of human skeletal cortical bone including phalanx bones of finger bone. This fracture bone seemingly insignificant among us, but it may lead to fine motor dysfunction and chronic problem (Prentice, 2006). Thus, the injury may not only limit the functionality of a finger’s usefulness in activities of daily living.

Fracture of finger bone can be categorized into malunion, nonunion and posttraumatic osteoarthritis (Ring, 2005). The fracture phenomena in cortical bone are accompanied by formation of microcrack or microcracks accumulation (O'Brien et al., 2005). Therefore, this study interested to investigate fracture phenomena in phalanx cortical of finger bone.

This study aims is to evaluate stress shielding induced by crack interaction using a simple model based on Linear Elastic Fracture Mechanics (LEFM). This simulation based on the determination of the Stress Intensity Factor (SIF) and the changes of stress shielding in different crack interval towards the human phalanx bone. The stress shielding effect usually occurs at two parallel edge cracks of finite body.

1.2 Problem Statement

Fingers allow us to perform specialized functions such as grasping a pen or manipulating small objects in our palm. When a finger bone is fractured, it can cause the whole of our hand to be out of alignment. However, bone fracture may leave permanent damage and a long recovery. Normal healing time for bone is about four to six weeks, whereas for small bones, it may even heal in as little as three weeks (Hajdas, 2009). This situation can be prevented if we understand the biomechanics of bone behavior. There are various types of biomechanical behavior of bone and foremost is shielding interaction.

Numerical simulation is a one of the methods that best to evaluate the shielding interaction as well as to predict its behavior under prescribed conditions and problems. Therefore, numerical simulation will be carried out in this project to understand the stress shielding induced by crack interaction consequently expedite the healing time of bone.

1.3 Research Objective

The objective of this study is to find the value of Stress Intensity Factor (SIF) ona human phalanx of finger bone within the variation of crack interval. This project required the use of Finite Element Analysis (FEA) for further analysis on the fracture behavior of double parallel edge cracks in phalanx bone. The two-dimensional model will be constructed and the crack interval on a human phalanx of finger bone will be altered using ANSYS software. Therefore, it can be concluded with four main objectives in this study, including:

1) To develop two-dimensional (2-D) model based on Linear Elastic Fracture Mechanics (LEFM) in continuum body.
2) To identify the effect of crack interval on Stress Intensity Factor (SIF).
3) To evaluate the stress shielding effect on the fracture behavior of double parallel edge cracks in phalanx bone.
4) To evaluate the crack interaction limit (CIL) and crack unification limit (CUL) on the fracture behavior of double parallel edge cracks in phalanx bone.

1.4 Research Scope

The study is focused in presenting the mathematical modeling of human phalanx cortical bone with biomechanical properties of the bone. This study needs to be acquainted with biomechanical properties of human phalanx cortical bone and basic principles of the finite continuum body. Through the stress singularity formulation, the study is limited to SIF determination at the crack tips. This calculation is done by numerical simulation in

ANSYS software. Finally, the values of SIF will indicate the stress intensity level at the crack tips.

1.5 Research significance

The significance of this research project is pinpoint to the interaction between cracks in cracked bone. The interaction is vital aspect, especially in the medical sector. The interaction of cracks indicates the human pain within the phalanx bone. As nowadays, several of implant artificial bone have been used extensively to replace fractured bone towards helping recover the pain in bone. Hence, if the unification limit is known, the pain can be reduced. In addition, this project also provides the maximum limit of crack interaction based on several parameters. These are needed because in real life situation, human phalanx bone has possibility getting completely damage if the parameters exceed certain values, for example; the load is applied too much on the bone.

1.6 Organization of the dissertation report

Chapter 1 begins with introduction of human bone. This chapter also explained about fracture of human bone including in phalanx bones of finger bone. The objectives of this study also included to explain about what will expect from this study.

Chapter 2 covers the literature review concerning the study on research and methodology that are related with this project or the past thesis and project.

Chapter 3 discussed on the methodology, where it would outlined the tools to run the project and the method used. This project can be described in three stages. The first stage, the two-dimensional (2-D) human phalanx model would be constructed in ANSYS APDL software. The second stage described on the finite element analysis (FEA) process. This stage involved the area that would be plotted on 2-D model before it was being mesh with suitable size of meshing and static uniaxial loading that was subjected on 2-D model. The third stage demonstrated upon the use of the finding in finite element.

Chapter 4 discussed on the results as well as the discussion. In this chapter, the obtained results were being discussed in details.

Chapter 5 was a summary on the project overall. The recommendation may be included for future study.

CHAPTER 2

LITERATURE REVIEW

2.1 Introduction

Bone is the main structural component of the body, providing an internal framework for the body. It also serves as levers for the muscles to pull on, subsequently cause movements in joints. The structural basis of bone can be classified into two types of macroscopic approach; compact bone (cortical bone) and spongy bone. Compact bone is a bone that contributes the majority of human weight due to its dense structure, lower surface area and lower porosity. Thus, compact bone is functional in supporting the human body and protecting the soft organ. In contrast, the spongy bone is less dense, has large bone space and has higher porosity. This makes the spongy bone prefer functioning in mobilization than supporting and protecting.

The average total of bones in adult human skeleton is about 206 bones, including phalanges bone of the human hand. Phalanges bone is a vital segment of the human body in helping human do many activities in their daily life. Therefore, the interest area in this project is on the phalanx bone of the human hand. The phalanx bone of human hand located at fingers. Generally, the fingers are formed by three phalanges; proximal, middle and distal phalanx. Figure 2.1 illustrated in details the structures of fingers hand.

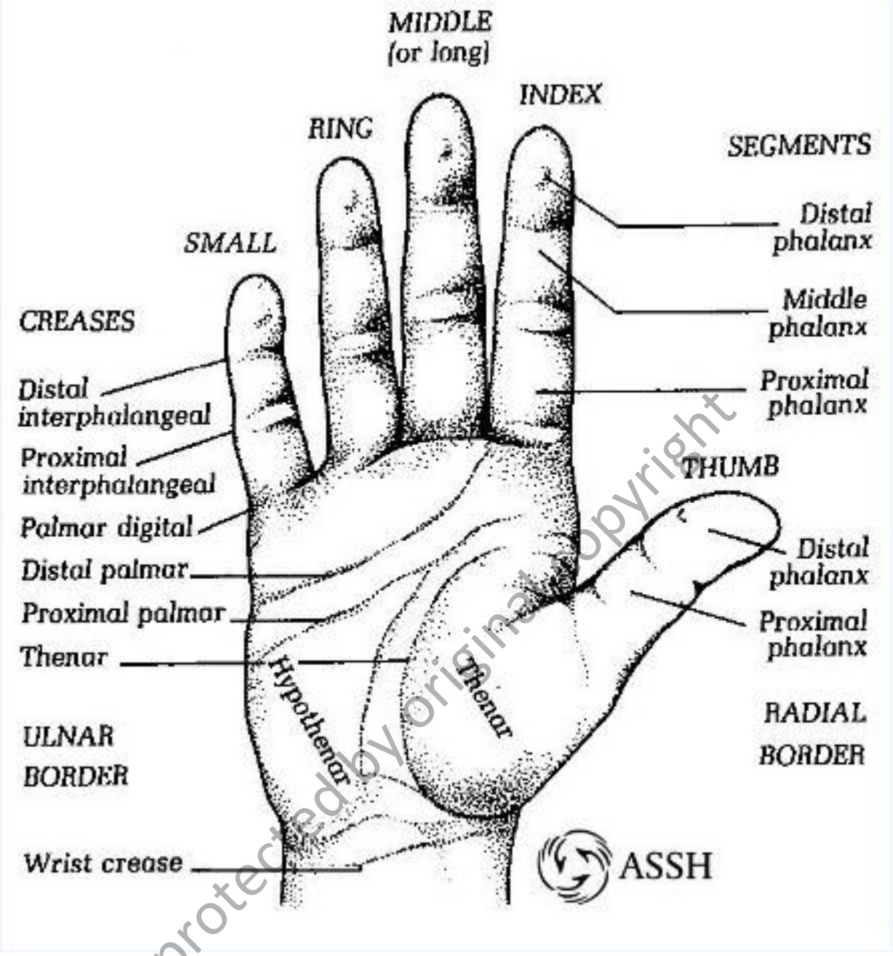

Figure 2.1: The structure of fingers hand (Seiler, 2002)

Human phalanx bones consist of a compact bone tissue cylinder which penetrated by medullar cavity and inside the medullar cavity is the bone marrow. Then, at the end of the bones are built by a compact substance thin layer outside and a spongy mass inside. This shows that human phalanx bone having a complex biological tissue. However, this type of bone can be modeled in the finite element analysis (Tarnita et al., 2005).

2.2 Fractures in bones

From 206 bones of the human skeleton, it can be classified into four groups; long bones, short bones, flat bones and irregular bones (Marieb, 2009). These bone classified is based on the shape of the bone.

Long bones relatively in long and slender shape. This type of bone usually situated in the arms, legs, thighs, palms, soles, fingers and toes. Then, short bones having cube-shaped which length and width is in same size. This short bone usually be found in the bone of the wrists (carpals) and bone of the ankle (tarsals).

Next are flat bones which having thin, flat surfaces and no marrow cavity structure. However, this type of bone has the spongy bone sandwiched between layers of compact bone.Also, this type of bone can be searched in sternum, ribs, scapula, clavical and the bones which shape the roof of the skull (parietal, frontal, temporal and occipital).Last are irregular bones that have the complexity of shape, notched or with ridges. This irregular bone commonly located in the facial bones, pelvic bones, heel bone and mandible.

Therefore, this study focused on long bones fractured of human phalanx bone (finger bone). In addition, long bone having particular features such as hard and dense bone. These features had provided strength, structure and mobility. The structure and fractures in long bones will be discussed in details in next subtopic.

2.2.1 Fractures in Long Bones

Phalanges bones (finger bone) can be defined as long bone. These types of bones can be categorized into two regions; diaphysis and epiphysis. The structure of long bones that connect diaphysis with epiphysis is known as metaphysis. The long bone fractured, sometimes may occur at a portion of metaphysis or sometimes at the diaphysis.

There are various types of fracture patterns occur at diaphysis such as intra-articular, transverse, oblique and spiral. In this study, transverse fractures were highlighted and these types of fractures usually related to angulate injuries or direct blows (Dent, 2008). Figure 2.2 illustrated in details the pattern of fracture diaphysis long bone.

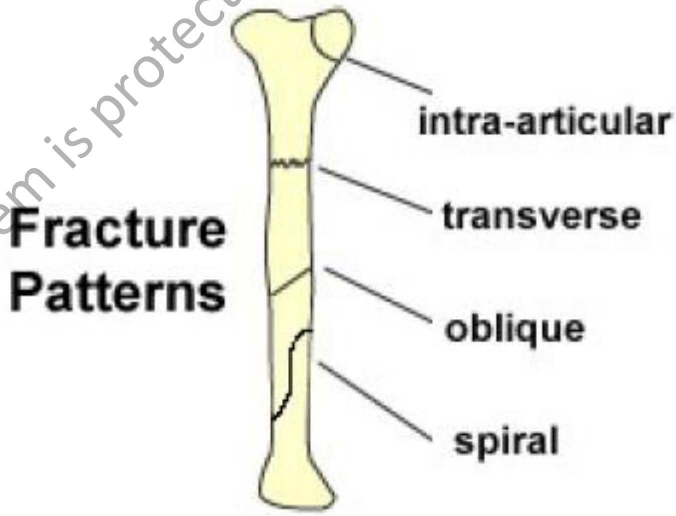

Figure 2.2: The patterns of fracture diaphysis long bone (Dent, 2008)

This transverse pattern can be related to the malalignment of phalanx bone. Malalignment can be formed on phalanx bone in three conditions such as angulated, displaced and rotated. These types of malalignment conditions had been depicted in details in Figure 2.3. Displaced condition had been focused on this study as this type of condition usually occur when the two bone ends are shifted on each other to different extents either in a forward/backwards direction or a sideways direction (Dent, 2008). Extensive works that had been done by other researchers on fractures in cortical bone will be discussed in next subtopics.

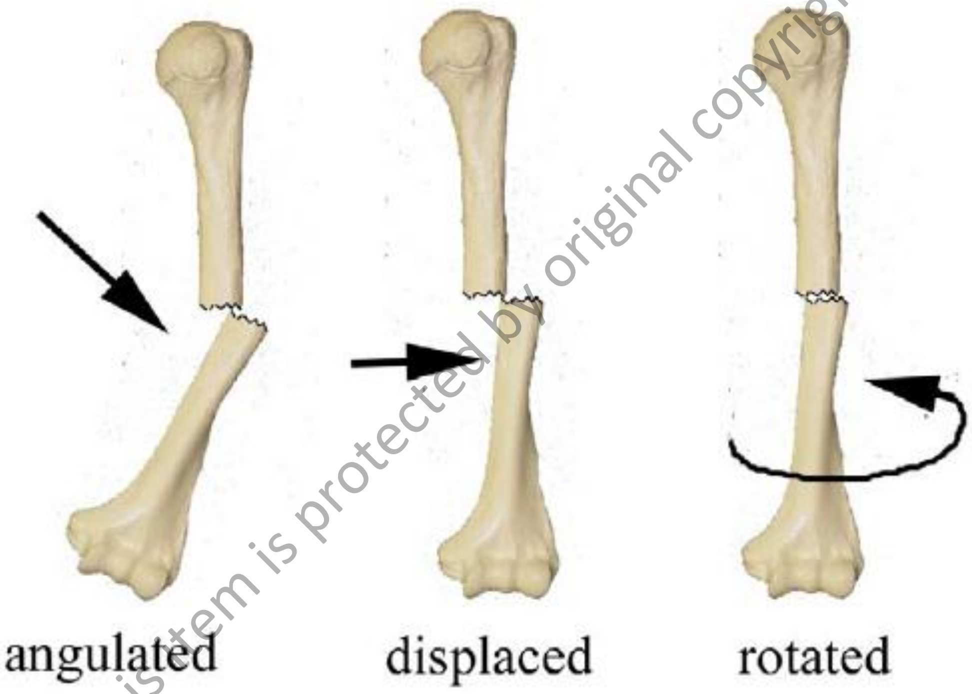

Figure 2.3: The types of malalignment condition on phalanx long bone (Dent, 2008).

2.2.2 Fractures in Human Cortical Bone

During the last decades, interaction of microcracks or fracture in human cortical bone attracts researchers' attention. There were extensive works in developing the most accurate methods to be applied.

Najafi et al. (2009) accomplished a study that adopted Green's function to formulate a system of singular integral equations for the general microcracks in vicinity of the osteon. Green's function is solution for the edge dislocations. This paper presented a two-dimensional micromechanical fiber-ceramic matrix composite material model that based on LEFM to evaluate the interaction between microcracks. This paper also studied the interaction between microcracks and the bone microstructure by understanding the effect of microstructural morphology and heterogeneity towards fracture behavior. The results show that when osteon is softer than interstitial tissue, the SIF is increased. For osteon is harder than interstitial tissue, the SIF is found to be decreased. It indicated that the SIF value depending on material mechanical properties. The SIF value also dependent on the configuration of microcracks either in the shape of stress amplification or in the shape of stress shielding.

Then, Shah et al. (2009) proposed two resin based dental composite materials; microhybrid and nanofill in this study to investigate significant fracture and toughening mechanism. This study used an R-curve approach to characterize the fracture behavior of two dental composites. An experiment was conducted in this study including sample preparation, R curve testing, flexural strength testing, double notched experiment, and fracture and toughening mechanism. Fracture and toughening mechanism was characterize by Scanning Electron Microscope (SEM). Results showed that microhybrid composite having a 20% higher mean flexural strength compared to nanofill composite. The fracture resistance of both composite increases with crack extension over ~1 mm of crack length.The higher rising R-curve of the microhybrid dental composite indicated higher fracture toughness and higher flexural strength but less scatter in flexural strength. Two different extrinsic toughening mechanisms were identified; crack deflection and crack bridging. Although Nanofill composite have the lower strength and toughness but this nanoparticle cluster is more effective at deflecting crack and imparting as the solid particles of the microhybrid.

Morais et al. (2010) conducted the Double Cantilever Beam (DCB) test in determining fracture toughness under pure mode I of cortical bone. A new data reduction scheme based on specimen compliance and crack equivalent concepts was introduced to overcome the difficulties in monitoring crack length of material. A cohesive zone model was used to simulate damage initiation and propagation, thus assessing the efficacy of the proposed testing method and data reduction scheme. Then, the DCB test was applied to evaluate the mode I fracture energy of hydrated and thermally dehydrated cortical bone tissue from the young bovine femur, in the tangential-longitudinal propagation system. Results showed that the fracture process zone of dehydrated bone is much less that in hydrated tissue. The interaction between water and collagen matrix contributes significantly to fracture process and cortical bone fracture behavior. The DCB test and the proposed data reduction scheme on the bone fracture characterization under mode I loading l was demonstrated efficiently in this study.

While, Ural et al. (2011) focused on traumatic fractures of the human cortical bone. The main objective of this study is to develop a computational approach based on cohesive finite element modelling in evaluating the effect of strain rate on both initiation and propagation toughness of human cortical bone. Two finite element modeling of compact tension was constructed, but in different dimensions. The first model includes the two dimensions (2-D) were simulated to evaluate the change in initiation and propagation fracture toughness with increasing strain rate. Second model involved three-dimensional (3-D) micro computed-tomography (μCT) which to assess the effect of porosity with