EXPERIMENTAL AND FINITE ELEMENT ANALYSIS OF FRACTURE BEHAVIOR OF FIBER REINFORCED PLASTIC

MOHD SUFFIAN B. MOHD THANI

Laporan ini diserahkan kepada Fakulti Kejuruteraan Mekanikal

sebagai memenuhi sebahagian daripada syarat penganugerahan

Ijazah Sarjana Muda Kejuruteraan Mekanikal (Struktur-Bahan)

Fakulti Kejuruteraan Mekanikal

Universiti Teknikal Malaysia Melaka

Mei 2008

"Saya akui bahawa saya telah membaca karya ini dan pada pandangan saya karya ini adalah memadai dari segi skop dan kualiti untuk tujuan penganugerahan Ijazah Sarjana Muda Kejuruteraan Mekanikal (Struktur-Bahan)"

Tandatangan : ....

Nama Penyelia : Pn Zakiah bt. Abd Halim

Tarikh : 15 Mei 2008

"Saya akui laporan ini adalah hasil kerja saya sendiri kecuali ringkasan dan petikan yang tiap-tiap satunya saya jelaskan sumbernya"

Tandatangan	: ....
Nama	: MOHD SUFFIAN B. MOHD THANI
Tarikh	: 15 Mei 2008

Projek Sarjana Muda ini didedikasikan khas buat ayah, ibu serta adik-adik tersayang di Sungai Besar, Selangor Darul Ehsan.

ACKNOWLEDGEMENT

First and foremost, ALHAMDULLAH I am grateful to ALLAH SWT on blessing in completing this project.

Mere words fail to express my immense thanks to my supervisors Mrs Zakiah bt. Abd Hakim, Mr Juzaila and Mr. Zaid b. Akop for their guidance and help in completing this report.

For people who offered an endless generosity and kindness especially for Mr. Rashdan and Mr. Ridwan from the mechanical department.

For my parents Mr. Thani b. Darus and Mrs. Faridah bt. Pilus for their encouragement and patience, a very special thanks for them.

Special Thanks to my friends and those who also supports my project.

ABSTRACT

This project is focused on analyzing fracture behavior (Mode I) of fiber reinforced plastic composite via experimental and finite element analysis simulation. The project will involved on developing a test specimen which is a glass fiber reinforced composite using epoxy resin as the matrix material. Three point bending test will be conducted on to the specimen (mode 1 fracture behavior). As for the finite element analysis simulation (Virtual Testing), Nastran Patran is used to model the test specimen and simulate the crack growth and determine the location of the cracks expected to grow. From both experiment and simulation, observation and analysis are conducted to compare the results of each method.

ABSTRAK

Projek ini menfokuskan kepada analisa kelakuan retak/patah (mod I) pada bahan komposit plastic diperkuat gentian dengan menggunakan kaedah eksperimen dan juga simulasi analisis unsure terhingga. Projek ini melibatkan penghasilan specimen ujikaji yang mana gentian kaca diperkuat komposit menggunakan bahan resin epoxy sebagai bahan matriks. Ujian 'Three Point Bending' akan dijalankan kepada specimen tersebut (mod I kelakuan retak/patah). Bagi simulasi analisis unsur terhingga (ujian visual), program simulasi Nastran Patran digunakan utk menghasilkan model specimen ujikaji dan menjalankan simulasi pembentukan retak/patah dan menentukan lokasi di mana retak/patah akan terbentuk. Daripada kedua-dua ujikaji dan simulasi yang telah dijalankan, pemerhatian dan juga analisis dilakukan bagi membandingkan hasil keputusan bagi kedua-dua kaedah.

TABLE OF CONTENT

LIST OF FIGURES

LIST OF TABLES

LIST OF SYMBOLS

SYMBOL	DESCRIPTION
Ef	Flexural Modulus
σ	Maximum Stress
S	Support Span
m	Slope of the load
Kq	Fracture Toughness

LIST OF APPENDICES

CHAPTER 1

INTRODUCTION

1.1 Background Study

Composite material usually consists of a matrix and a reinforcing constituent. The matrix is often soft and ductile compared to the reinforcement, but this is not always the case. Various types of reinforcements are possible including continuous fibers, chopped fibers, whiskers, flakes and particulates [19].

When a polymer matrix is combined with strong, high modulus reinforcement, the resulting material can have superior strength/weight and stiffness/weight ratios compared to steel and aluminum. Continuous fiber-reinforced plastics tend to give the best overall performance (compared to other types of polymer composites), but can also exhibit troubling fracture and damage behavior.

A variety of fiber-reinforced polymer composites are commercially available. The matrix material is usually a thermoset polymer (i.e., an epoxy), although thermoplastic composites have become increasingly popular in recent years. Polymers reinforced by continuous graphite or Kevlar are intended for high performance applications such as fighter planes, while fiber glass is an example of a polymer composite that appears in more down-to-earth applications. The latter material consists of randomly oriented chopped glass fibers in a thermoset matrix.

Figure 1.0a illustrates the structure of a fiber reinforced composite. Consider a single ply (fig 1.1a). The material has high strength and stiffness in the fiber direction, but has relatively poor mechanical properties when loaded transverse to the fibers. In other words, the strength and stiffness are controlled by the properties of the matrix. When the composite is subjected to biaxial loading, several plies with differing fiber orientations can be bonded to form a laminated composite. The individual plies interact to produce complex elastic properties in the laminate. The desired elastic response can be achieved through the appropriate choice of the fiber and matrix material, the fiber volume, and the lay-up sequence of the plies, T.L Anderson (2000).

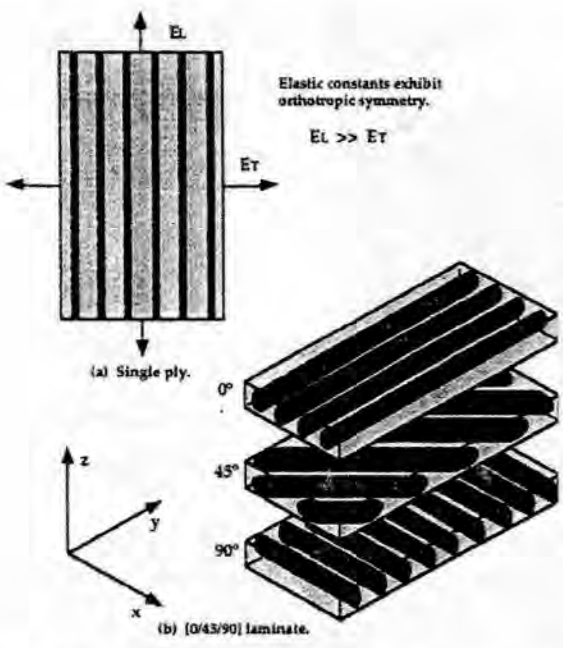

Figure 1.0 a structure of fiber reinforced composite.

(Source: T.L Anderson (2000))

1.2 Problem Statement

Nonmetals, like metals are not immune to fracture, but compare to fracture of metals, research into the fracture behavior of non-metals (i.e. composites) is still new. Much of the necessary theoretical framework is not yet fully developed for nonmetals, and there are many situations where fracture mechanics concepts that apply to metals have been misapplied to nonmetals.

Conventional fracture mechanics methodology assume a single dominant crack that grows in a self similar fashion; i.e. the crack increases in size (either through stable or unstable growth), but its shape and orientation remain the same. Fracture of a fiber-reinforced composite, however, is controlled by numerous micro cracks distributed throughout the material, rather than a single macroscopic crack. There are situations where fracture mechanics is appropriate for composites, but it is important to recognize the limitations of theories that were intended for homogeneous materials.

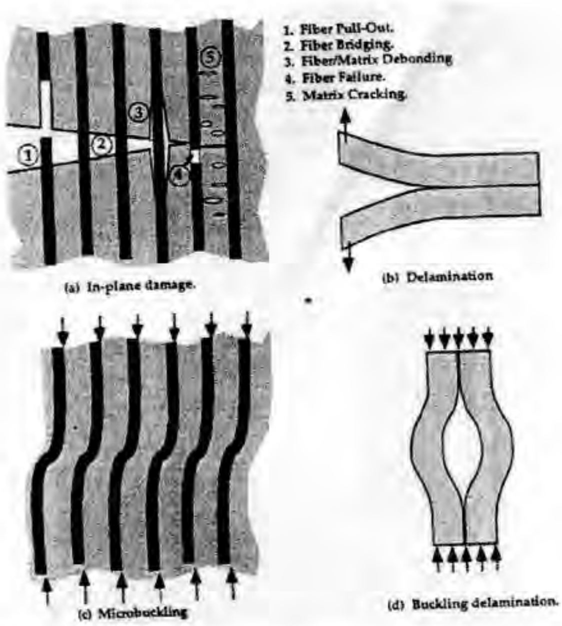

Figure 1.2 failures in fiber reinforced composites.

(Source: T.L Anderson (2000))

Figure 1.2 illustrates various failure mechanisms in fiber-reinforced composites. One advantage of composite materials is that fracture seldom occurs catastrophically without warning, but tends to be progressive, with sub-critical damage widely dispersed through the material. Tensile loading can produce matrix cracking, fiber bridging, fiber rupture, fiber pullout, and fiber/matrix debonding. Ultimate tensile failure of a fiber reinforced composite often involves several of these mechanisms. Out-of-plane stresses can lead to delamination, because the fibers do not contribute significantly to strength in this direction. Compressive loading can produce microbuckling of fibers; since the polymer matrix is soft compared to the fibers, the fibers are unstable in compression. Compressive loading can also lead to microscopic delamination buckling, particularly if the material contains a pre-existing delaminated region.

1.3 Problem Identification

In problem statement, some of the difficulties in applying fracture mechanics to fiber reinforced plastic have been explain. Since there is a less rigorous framework to describe fracture behavior in composites has led a number of quantitative approaches to characterize toughness and one of them is interlaminar fracture.

Interlaminar fracture is one of the few instances where fracture mechanisms formalism is applicable to fiber reinforced composites on a global scale. A zone of delamination can be treated as a crack; the resistance of the material to the propagation of this crack is the fracture toughness. Since the crack typically is confined to the matrix material between plies, continuum theories is applicable and and the crack growth is self similar. Interlaminar fracture/cracking or delamination can occur under three basic modes as shown on figure 1.4.1a; opening or peel mode (mode I), forward or sliding shear mode (mode II), or tearing mode (modeIII), or under combination mode (i.e. mode I & mode II).

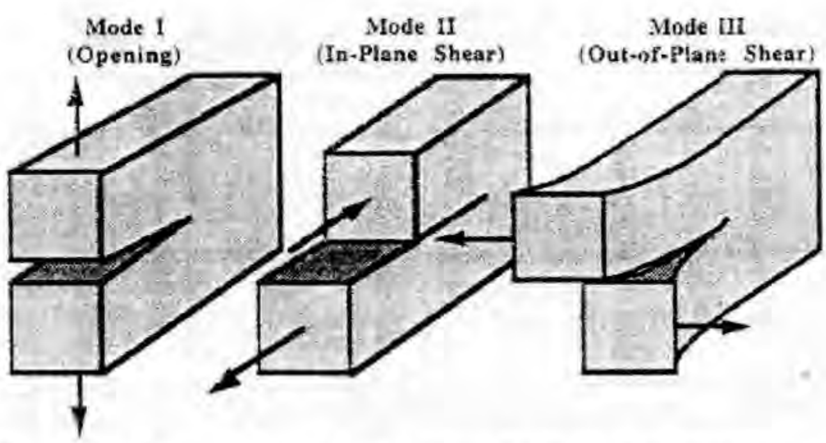

Figure 1.3 three modes of interlaminar fracture in composite material.

There are three common specimen configurations for interlaminar fracture toughness; double cantilever beam (DCB) specimens are the most common configuration for this test. One of the advantages of DCB that is it permits measurement/analysis of Mode I, Mode II, or mixed mode fracture toughness. The other configurations are the end notched flexure (ENF) specimen. It has the same geometry as the DCB specimen, the only different is that the specimen is loaded in three (3) point bending/flexure test, which impose to Mode II displacement of the crack faces (in-plane shear).Analogous method can be applied to obtain Mode I or Mode II analysis to other specimen configuration; examples are the three (3) point bending test/flexure testing will be used in this project to determine the fracture behavior of fiber reinforced composites.

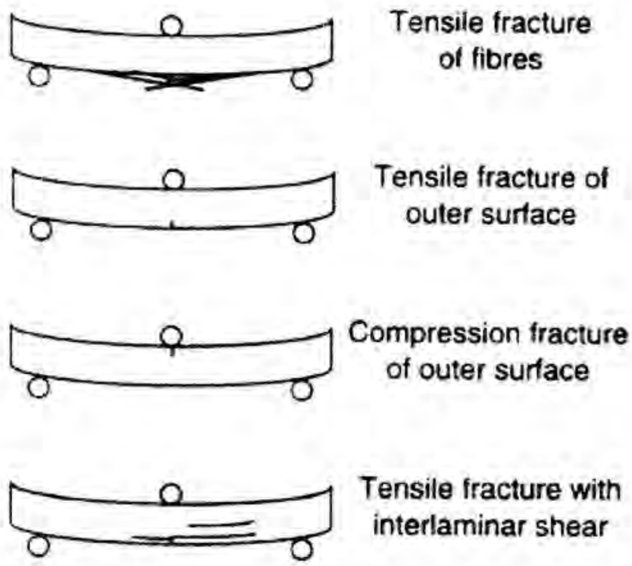

Figure 1.4 Type of 3 point bending tests proposed for this project.

1.4 Objectives

For the experimental and finite element analysis of fracture behavior of fiber reinforced plastics, few objectives and scope has been listed as guidance during planning and conducting the project. The **highlight** objectives below are the main objective of this project which is a finite element analysis simulation using Nastran Patran Software and the comparison of result between experimental (three point bending test) and simulation result. Below are the objectives of the project:

- To carry out three points bending test on the fiber reinforced plastic specimen.
- To model a finite element model for crack growth simulation in Pro Engineer/Solidwork/Patran software.
- To simulate the crack growth and determine the location of the cracks are expected to grow.
- **To analyze the fracture behaviors of the fiber reinforced plastic and compare the experimental and simulation results.**

In this study, glass fiber reinforced epoxy will be used as the test specimen in three point bending test. After the lamination process, the test specimens will undergo three point bending test to characterize its mechanical properties and analyze the fracture behavior. Two Dimensions simulation will be demonstrated where the test specimen models are based on the actual dimension and properties of the composite.

1.5 Scope

The project is involved the preparation of the laminated specimens (test piece) in a standard dimension (laboratory scale), a flexure/bending test, simulation and analysis of both method. As mention before, the experiment and simulation will start/conduct in next semester (PSM2). Below are the scopes of the project:

- Laboratory scale for laminate polymer matrix composite.
- Laboratory scale for three point bending.
- Mode 1 fracture behavior.
- Model the specimen using Pro Engineer/**Solidwork/Patran Educational**/Version software.
- Detailed modeling strategies, pre-processing, and post-processing steps in **Nastran-Patran.**

1.6 Progress Planning

In PSM 1, literature review is carried out to understand the problem statement, to identify the problem and project planning. By understanding the problem, it helps on deciding the scope of study where the material and test specimen are choose which a fiberglass/epoxy composite. In addition, previous work related to the scope of study is also reviewed where it gave some ideas and guidance about the methodology of the experiment and the finite element analysis for 2 dimension simulation. The objective/scope for PSM I is done.

For PSM II, the preparation for test specimen and 3 point bending test are carried out. 8 specimen will be provided and these specimen will undergo a 3 point bending test for Mode I fracture toughness to analyze the fracture behavior. The 2 dimensional finite element analysis simulation also will be conducted In PSM II, 2D model and simulation will be demonstrated to analyze and determine the location of crack growth.

The result for fracture behavior fiber reinforced plastic will be analyze and compare the result between experimental and simulation.