



UNIVERSITI PUTRA MALAYSIA

***FINITE ELEMENT ANALYSIS OF COHESIVE ELEMENT
FOR GLASS EPOXY COMPOSITE FUSELAGE MODEL***

NG WEI SIM

FK 2011 2

**FINITE ELEMENT ANALYSIS OF COHESIVE ELEMENT FOR GLASS
EPOXY COMPOSITE FUSELAGE MODEL**



NG WEI SIM

**MASTER OF SCIENCE
UNIVERSITI PUTRA MALAYSIA**

2011

**FINITE ELEMENT ANALYSIS OF COHESIVE ELEMENT FOR GLASS
EPOXY COMPOSITE FUSELAGE MODEL**

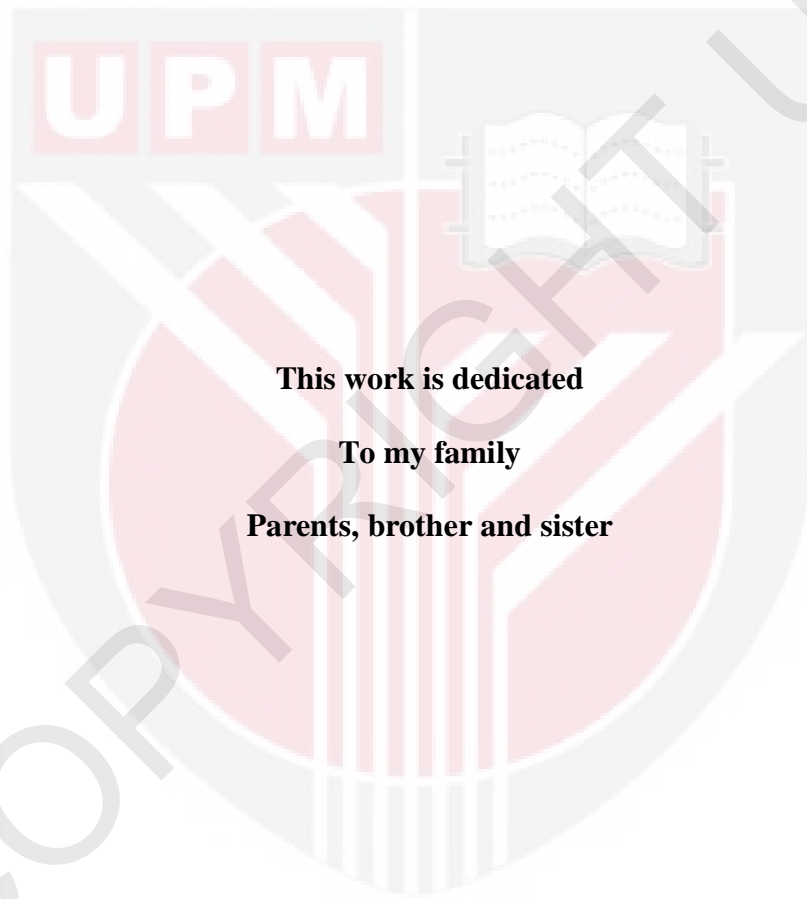


By

NG WEI SIM

**Thesis Submitted to the School of Graduate Studies Universiti Putra Malaysia in
Fulfilment of the Requirement for the Degree of Master Science**

January 2011



This work is dedicated

To my family

Parents, brother and sister

Abstract of thesis presented to the Senate of Universiti Putra Malaysia in fulfilment of the requirement for the degree of Master Science

FINITE ELEMENT ANALYSIS OF COHESIVE ELEMENT FOR GLASS EPOXY COMPOSITE FUSELAGE MODEL

By

NG WEI SIM

January 2011

Chairman: Faizal Mustapha, PhD

Faculty: Engineering

In this research, a numerical simulation on a newly introduced fabrication miniature composite fuselage structure - a glass epoxy composite laminated with adhesively bonded butt joint under axial compression loading is presented. A FEA via ABAQUS/Explicit is utilized to capture the complete compressive response to predict the crushing behaviour and mechanical strength from initial compression loading up to the final failure is demonstrated. A woven C-glass fiber/epoxy 200 g/m² composite laminated with the orthotropic elastic material properties is modeled as continuum composite lay-up in the proposed numerical model. The adhesively bonded joint progression is modeled using cohesive elements technology that allows the correct accounting for the energy involved in the crushing process and investigates the capability of the bonded joint to withstand the axial crushing impact from debonding failure. An experiment of Double Cantilever Beam (DCB) according to the ASTM

Standard D5528 is performed to determine the adhesive Mode I critical toughness. The crushing load and collapse modes under axial compression loading from FEA results indicate good correlation with the experimental test. The discrepancy of the peak load is found to be at 1%.



Abstrak tesis yang dikemukakan kepada Senat Universiti Putra Malaysia sebagai memenuhi keperluan untuk ijazah Master Sains

ANALISIS UNSUR TERHINGGA PADA UNSUR KOHESIF UNTUK MODEL FUSLAJ KOMPOSIT EPOKSI KACA

Oleh

NG WEI SIM

January 2011

Pengerusi: Faizal Mustapha, PhD

Fakulti: Kejuruteraan

Dalam kajian ini, simulasi berangka pada fabrikasi baru untuk miniatur komposit fuslaj struktur - komposit epoksi kaca lamit dengan ikatan pelekat akibat beban mampatan paksian ditunjukkan. Analisis Unsur Terhingga (FEA) dengan menggunakan ABAQUS/Eksplisit untuk mengkaji tindak balas mampatan bagi meramalkan perilaku hancuran dan kekuatan mekanik dari mampatan awal sehingga kegagalan akhir dikenal pasti. Gentian kaca C/epoksi 200 g/m² komposit berlapis belitan yang bersifat elastik orthotropik dimodelkan sebagai kontinum komposit dalam model berangka yang dicadangkan. Ikatan pelekat dimodelkan dengan menggunakan teknologi unsur kohesif untuk pengiraan tenaga yang tepat dalam proses mampatan dan menyiasat kemampuan ikatan pelekat untuk menahan kesan paksian mampatan daripada kegagalan. Ujian mengikut penilaian ASTM D5528 Double Cantilever Beam (DCB) dijalankan untuk menentukan pelekat kritis kekuatan mod I. Daya dan mod hancuran mampatan paksian

hasil daripada keputusan Analisis Unsur Terhingga menunjukkan korelasi yang baik dengan eksperimen. Perbezaan daya puncak didapati sebanyak 1%.



ACKNOWLEDGEMENTS

I would like to express my deepest gratitude to Dr. Faizal Mustapha for his encouragement, valuable advice, and guidance through my years as a master student. It is a pleasure and an honour to be his student.

I would like also to express my sincere gratitude and deep thanks to my supervisory committee member, Dr. Rizal Zahari for his kind assistance, advice, encouragement, and suggestions throughout this work and during the preparation of this thesis. I would like to thank Mr. Saifol for helping me to carry out the experiment. Last but not least, thanks to all who have helped me in this endeavor.

Ng Wei Sim

I certify that a Thesis Examination Committee has met on 28 January 2011 to conduct the final examination of Ng Wei Sim on her thesis entitled “Finite Element Analysis Of Cohesive Element For Glass Epoxy Composite Fuselage Model” in accordance with Universities and University College Act 1971 and the Constitution of the Universiti Putra Malaysia [P.U.(A) 106] 15 March 1998. The committee recommends that the student be awarded the Master of Science.

Members of the Thesis Examination Committee were as follows:

Mohd Ramly bin Ajir, PhD, Lt, Col, (R)

Associate Professor
Faculty of Engineering
Universiti Putra Malaysia
(Chairman)

Mohd Khairol Anuar bin Mohd Ariffin, PhD

Senior Lecturer
Faculty of Engineering
Universiti Putra Malaysia
(Internal Examiner)

Mohd Sapuan bin Salit @ Sinon, PhD

Professor
Faculty of Engineering
Universiti Putra Malaysia
(Internal examiner)

Zainal Ariffin Ahmad, PhD

Professor
School of Materials and Mineral Resources Engineering
Universiti Sains Malaysia
(External Examiner)

BUJANG BIN KIM HUAT, PhD

Professor and Deputy Dean
School of Graduate Studies
Universiti Putra Malaysia

Date: 24 March 2011

This thesis was submitted to the Senate of Universiti Putra Malaysia and has been accepted as fulfilment of the requirement for the degree of Master of Science. The members of the Supervisory Committee were as follows:

Faizal Mustapha, PhD

Senior Lecturer
Faculty of Engineering
Universiti Putra Malaysia
(Chairman)

Rizal Zahari, PhD

Senior Lecturer
Faculty of Engineering
Universiti Putra Malaysia
(Member)

HASANAH MOHD GHAZALI, PhD

Professor and Dean
School of Graduate Studies
Universiti Putra Malaysia

Date:

DECLARATION

I declare that the thesis is my original work except for quotations and citations which have been duly acknowledged. I also declare that it has not been previously, and is not concurrently submitted for any other degree at Universiti Putra Malaysia or other institutions.

The logo of Universiti Putra Malaysia (UPM) is a shield-shaped emblem. At the top left, the letters 'UPM' are written in white on a red rectangular background. The shield is divided into four quadrants by a white cross. The top-right quadrant contains an open book. The bottom-left quadrant features a stylized white 'Y' shape. The bottom-right quadrant contains a stylized white 'M' shape. The background of the shield is a light red color. A horizontal line is drawn across the middle of the shield, passing through the text 'NG WEI SIM'.

NG WEI SIM

Date: 23 January 2011

TABLE OF CONTENTS

	Page
ABSTRACT	iii
ABSTRAK	v
AKNOWLEDGEMENT	vii
APPROVAL	viii
DECLARATION	xi
LIST OF TABLES	xiii
LIST OF FIGURES	xiv
NOMENCLATURE	xx
CHAPTER	
1. INTRODUCTION	
1.1 Research Background	1
1.2 Problem Statement	3
1.3 Research Hypothesis	4
1.4 Research Objectives	5
1.5 Scopes of Work	6
1.6 Significance of The Study	6
1.7 Thesis Organization	7
2. LITERATURE REVIEW	
2.1 Filament Winding	8
2.2 The Combined Mould Technique	10
2.3 Crushing Analysis	11
2.4 Finite Element Analysis (FEA) Softwares	13
2.5 FEA on Composite Materials	14
2.5.1 Creating Solid Composite Lay-ups [26-27]	15
2.5.2 Creating Conventional Shell Composite Lay-ups [26-27]	16
2.5.3 Creating Continuum Shell Composite Lay-ups [26-27]	16
2.6 FEA on Adhesive Joint	18
2.6.1 Fracture Mechanic and Damage Zone Model (cohesive)	19
2.6.2 VCCT	20
2.6.3 Cohesive Zone Model	21
2.6.4 Traction Separation Law for FEA [26-27]	25
2.6.5 Initial Elastic Stiffness	27
2.6.6 Damage Initiation: Interface Strength	30
2.6.7 Damage Evolution: Fracture Toughness	32
2.7 Experimental Techniques of Mode I - Mode III	36
2.8 Cohesive Element in Modelling	37
2.9 Summary	40
3. FEA METHODOLOGY	
3.1 Introduction	41

3.2 Problem Description	42
3.3 Preprocessing-creating The FEA Model Via ABAQUS/Explicit	44
3.4 Convergence Study	48
3.5 Summary	50
4 FEA RESULTS AND DISCUSSION	
4.1 Results	51
4.2 Discussion	51
4.3 Summary	54
5 EXPERIMENTAL TEST VALIDATION	
5.1 Specimen Preparation	55
5.2 Quasi-static Crushing Test	59
5.3 Quasi-static Crushing Results	59
5.4 FEA Validation of Fuselage Model	62
5.5 Summary	63
6 CONCLUSIONS AND RECOMMENDATIONS	
6.1 Conclusions	64
6.2 Recommendations for Further Work	65
REFERENCES	66
APPENDIX A–Material Parameters Definition	71
APPENDIX B–Double Cantilever Beam (ASTM Standard D5528-01)	72
APPENDIX C–Preprocessing for Parts Definition	89
APPENDIX D–Preprocessing for Meshing Technique	96
APPENDIX E–Preprocessing for Material Definition	109
APPENDIX F–Preprocessing for Assembly Level	118
APPENDIX G–Preprocessing for Step Definition	119
APPENDIX H–Preprocessing for Field Outputs Definition	122
APPENDIX I–Preprocessing for History Outputs Definition	124
APPENDIX J–Preprocessing for Interaction Definition	127
APPENDIX K–Preprocessing for Constraint Definition	129
APPENDIX L–Preprocessing for Boundary Condition Definition	133
APPENDIX M–ABAQUS/Explicit Input File	136
APPENDIX N–Preprocessing for X–Y Data Plot	141
BIODATA OF STUDENT	143