STUDY BEHAVIOUR OF DIFFERENT ELEMENTS IN ANSYS FINITE ELEMENT SOFTWARE

ZAIDI BIN ZOLKEFLE

UNIVERSITI TEKNIKAL MALAYSIA MELAKA

SUPERVISOR DECLARATION

"I hereby declare that I have read this thesis and in my opinion this thesis is sufficient in terms of scope and quality for the award of the degree of Bachelor of Mechanical Engineering (Structure and Materials)"

Signature : ……

Supervisor's Name : PROF. MADYA AHMAD RIVAI

Date : ……

STUDY THE BEHAVIOUR OF DIFFERENT ELEMENTS IN ANSYS FINITE ELEMENTS SOFTWARE

ZAIDI BIN ZOLKEFLE

This Thesis Is Submitted In Partial Fulfillment of Requirement for the Bachelor Degree of Mechanical Engineering (Structure & Material)

\

Faculty of Mechanical Engineering

Universiti Teknikal Malaysia Melaka

JUNE 2015

DECLARATION

"I hereby declare that the work in this thesis is my own except for summaries and quotations which have been duly acknowledgement"

Signatures : ……

Name of Authors : ZAIDI BIN ZOLKEFLE

Date : ……

Especially for my father, Zolkefle Bin Haron and my mother,
Zarena Bte. Abd. Rahman

ACKNOWLEDGEMENT

First of all, I would like to give my sincere thanks to my supervisor, Professor Madya Ahmad Rivai, for his guidance, inspiring motivation and helps during the supervision of this project. I have been very fortunate to be the chosen student under his guidance. His constant voice of advices and constructive criticism, especially in writing this thesis has been my source of inspiration.

I would like to thanks my parents for give me a moral support and spirit to complete my final year project report. I would also like to acknowledge with much appreciation to staff of Mechanical Laboratory, who gave the permission to conduct the experiment for this project. They have given so much cooperation to me during three-point bending test.

Lastly, I would like to say thank you to all Mechanical Faculty's staff especially Jawatankuasa Projek Sarjana Muda (JKPSM) which have provided useful guidance and lots of information in order to write a good final thesis and some tips in preparation for seminar presentation.

ABSTRACT

Finite Element Analysis (FEA) is widely used in engineering field, especially in aircraft and structure engineering. Generally, it is a numerical method for solving engineering problems with complicated geometries, loadings and material properties. There are software that could be used to model and analyse FEA problems, such as Abaqus. ANSYS software is usually used to perform analysis on Computational Fluid Dynamics (CFD. The objective of this project is to study and investigate the capability of ANSYS in modelling and analysing structural problems by using different elements which are one-dimensional (1D), two-dimensional (2D) and three-dimensional (3D) elements. The validation and verification of the result will be done by comparison of analysis from ANSYS with experimental data, theoretical calculation and comparing the analysis result from other software. From the analysis, the distinction of the result from ANSYS with analysis of other software, theoretical calculation and experimental data was not too high with the percentage of error of 1 percent to 10 percent. So, ANSYS can be an effective platform to carry out structural modelling and analysing in Finite Element Analysis.

ABSTRAK

Analisis Unsur Terhingga (FEA) telah digunakan secara meluas dalam bidang kejuruteraan, terutamanya dalam kejuruteraan pesawat dan kejuruteraan struktur. Secara ammya, ia adalah satu kaedah berangka bagi menyelesaikan masalah kejuruteraan dengan geometri yang rumit, bersama beban dan sifat bahan yang tertentu. Terdapat beberapa perisisan komputer yang boleh digunakan untuk memodel dan menganalisa masalah FEA, seperti ABAQUS. Perisian ANSYS biasanya digunakan secara meluas untuk melakukan analisis berkenaan masalah Pengkomputeran Dinamik Bendalir (CFD). Projek ini bertujuan untuk mengkaji dan mempelajari keupayaan perisian ANSYS dalam permodelan dan menganalisis masalah struktur dengan menggunakan elemen yang berbeza iaitu elemen satu dimensi (1D), elemen dua dimensi (2D) dan elemen tiga dimensi (3D). Pengesahan dan pengukuhan akan keputusan yang telah dianalisis menggunakan perisian ANSYS akan dibandingkan dengan keputusan dariapada ujikaji, pengiraan teori dan perbandingan bersama hasil analisis daripada perisisan yang lain. Merujuk kepada hasil analisis, perbezaan keputusan perisian ANSYS dengan platfom yang berbeza tidak terlalu tinggi dengan peratusan perbezaan hanya dalam 1 sehingga 10 peratus. Jadi, ANSYS boleh menjadi platfom yang berkesan untuk menjalankan permodelan dan analisis terutama sekali dalam Analisis Unsur Terhingga (FEA).

CONTENT

LIST OF TABLES

LIST OF FIGURES

LIST OF GRAPHS

NOMENCLATURE

FE	Finite Element
FEA	Finite Element Analysis
FEM	Finite Element Method
DSA	Direct Stiffness Method
MSA	Matrix Stiffness Method
CAD	Computational Aid Diagram
CFD	Computational Fluid Dynamics
GUI	Graphics User Interface
APDL	ANSYS Parametric Design Language
Emag	Electromagnetic
Props	Properties
Prosproc	Proprocessor
1D	One-Dimensional
2D	Two-Dimensional
3D	Three-Dimensional
3PB	Three Point Bend
m	Meter
N	Newton
k	Kilo (X 10^3)

Pa	Pascal
G	Giga (X 10^9)
M	Mega (X 10^6)
σ	Stress
σ_x	Normal Stress
E	Modulus of Elasticity
σ_m	Moment of Stress
I	Moment of Inertia
ε	Strain
ε_x	Normal Strain

APPENDIX

CHAPTER 1

INTRODUCTION

1.1 Background of Project

Finite Element Analysis (FEA) initially developed in 1943 by R. Courant, whom responsible on THE RITZ method by relation of numerical analysis and minimization variation calculus to obtain approximate solutions to vibration systems (Khairul S. & Mariani Idroas, 2006). Chandrupatla & Belegundu (2012) state that for expositions convenience, structural "Finite Element" may be divided into four generations that last from 10 to 15 years. There are some evolutions of FEA of Matrix Structural Analysis (MSA) into Direct Stiffness Method (DSM) from 1934 to 1970.

Started from the writer, M. J. (Jon) Turner at Boeing over 1950 to 1962, he generalized and perfected the DSM and forcefully got Boeing to commit resources to it while other companies were mired into the Force Method. Then, in addition to Turner, major contribution to current practice include, B. M. Irons, inventors of ISO-parametric models, shape functions, and frontal servers, R. J. Melosh, who recognize the Rayleigh-Ritz link and synchronized the variation deviation of stiffness elements, and E. L. Wilson, who developed the first open source FEM software (Boeraeve P., 2010).

Then, M. J. Turner, R. W. Clough,, H. C. Martin, and L. J. Topp established a broader of numerical analysis papers in "STIFFNESS AND DEFLECTION OF COMPLEX SRTUCTURES". Early 70's, FEA was generally used in automotive, defense, aeronautics and nuclear industries. With the increasing of technologies level, FEA has been developed to an incredible precision for all kinds of parameters. FEM has become the most powerful tool for the numerical solution of a wide range of engineering problems. It can analysis in structures of heat flux, fluid flow, magnetic flux and complex problems (Chandrupatla and Blegundu, 2012).

Figure 1.1 shows the example of analysis by using Finite Element Analysis (FEA) on three-point bending test. It indicate the stress distribution towards the whole beam.

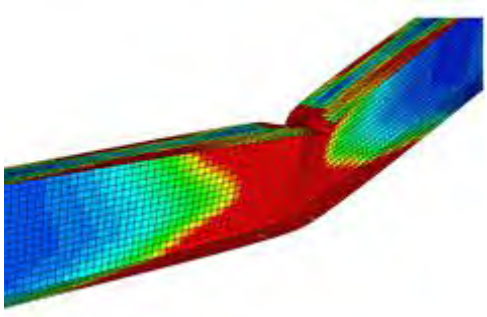

Figure 1.1: 3 Point Bend Analysis
(Source: Erickson, et. Al (2010))

Nowadays, Finite Elements Analysis is used to express the mechanical behaviour of a structure in engineering problems. There are three types of elements in finite element that are 1D (one-dimensional), 2D (two-dimensional) and 3D (three-dimensional) elements. For 1D, it is the simplest element, although lacking in their ability to make modelling on complex structure. 1D commonly uses as a line modelling to analyse any engineering problems with simple boundary condition and material properties. 2D elements and 3D elements provide more details and even more sophisticated results which require more complexity modelling on the structures. But,

the ability to gain the results on the Finite Elements also depends on the structures of the elements and the capability of the computer itself to read the element structures.

1.2 Problem Statement

There are many engineering software that has the ability to model or analyze any engineering problems. In performing structural analysis, especially for Finite Element Analysis, there are software that capable to conduct modelling and analyzing such as MSC Patran Nastran, ABAQUS, and ANSYS. But, in the industries, most of them adopt MSC Patran-Nastran as the platform to solve structural problems. The application of ANSYS software mostly covers on the Computational Fluid Dynamic (CFD) analysis.

For the exertion of ANSYS in finite element analysis only establish in late 1970's where the initial product was to analyze the simulation of fluid flow in engineering. There have been a deficiency of validation and verification of ANSYS especially in finite element. This project therefore will review the capability of ANSYS software to perform the modelling and analyzing of finite element structures.

1.3 Objective

To model appropriately the structural analysis problem in Finite Element Analysis (FEA) by using ANSYS Software.