



**FINITE ELEMENT ANALYSIS OF HIGH SUBSONIC AIR TARGET
DRONE WING STRUCTURE (HIGH SWEEP CONFIGURATION)**

FAIRUL ADZREEN BIN ABD WAHAB

(2005607289)

A thesis submitted in partial fulfillment of the requirement for the award of Bachelor
of Engineering (Hons.) Mechanical

**Faculty of Mechanical Engineering
Universiti Teknologi MARA (UiTM)**

MAY 2009

ACKNOWLEDGEMENT

First and foremost, Alhamdulillah and thanks to Allah S.W.T, the Almighty for giving me the strength and motivation to complete this research and thesis excellently.

I would like to express my sincere gratitude to my advisor Prof. Dr. Ir. Wahyu Kuntjoro for the continuous support of this research, for his patience, motivation, enthusiasm, and immense knowledge. His guidance helped me in all the time of research and writing of this thesis. I could not have imagined having a better advisor and mentor for my Degree study. I also want to express my deep appreciation to all lecturers, supporting staff and students of Faculty of Mechanical Engineering for their contribution during my Degree study.

Last but not the least, I would like to thank my family; my parents Abd Wahab Bin Mohamad and Kasiam Bte Omar, for giving birth to me at the first place and supporting me spiritually throughout my life.

ABSTRACT

In this project, finite element analysis (FEA) of the swept wing structure of an Unmanned Aerial Vehicle (UAV) for target drone purpose was performed. This project is very significant as it helps us to predict the behavior of a wing structure under a certain condition using FEA software MSC PATRAN and NASTRAN. For the purpose of finite element modeling, the CATIA file of the wing structure was imported into MSC PATRAN. The structure was then meshed using combination of CQUAD4 and TRIA3 elements. Aluminum alloys such 2024-T3 and AISI 4140 Steel with various thicknesses are employed for the wing structure sustaining Maximum Take-Off Weight of 250 kg under 4G loading condition. It was found that the maximum von Mises stress occurs on the vicinity of the connection between the wing and the fuselage, meanwhile the maximum displacement occurs at the tip of the wing structure. The results of the analysis were then checked using manual analysis to find the stress on a specific point on the wing structure and compared to the computational result. The structure will fail if stress is greater than the yield strength of the material used. These have proved that by using finite elements analysis, the behavior of structures can be predicted before they are to be produced.

TABLE OF CONTENTS

CONTENTS		PAGE
	ACKNOWLEDGEMENT	i
	ABSTRACT	ii
	TABLE OF CONTENTS	iii
	LIST OF TABLES	vi
	LIST OF FIGURES	vii
	LIST OF ABBREVIATIONS	x
CHAPTER I	INTRODUCTION	
1.1	Project Background	1
1.2	Objective	2
1.3	Significance of the Project	2
1.4	Scope of Work	3
CHAPTER II	LITERATURE REVIEW	
2.1	Finite Element Analysis	4
2.2	History of Finite Element Method	6
2.3	Unmanned Aerial Vehicle (UAV)	6
2.4	Swept Wing	7
2.5	Loads on Structural Components	8

2.6	Maximum Take-off Weight and Airworthiness	10
2.7	Past Researchers	10

CHAPTER III METHODOLOGY

3.1	Overview	13
3.2	Literature Review and Learning MSC SOFTWARE	15
3.3	Importing the Wing Structure	15
3.4	Building New Geometry in MSC PATRAN	15
3.5	Material Definition	17
3.6	Mesh Generation	
3.7	Properties Definition	20
3.8	Boundary Conditions and Load	21
	3.8.1 Load Exerted on the Wing Structure	22
	3.8.2 Elliptically Distributed Load	22
	3.8.3 Constraints	27
3.9	Obtaining Solution and Review	29
3.10	Results validation	30
3.11	Modification of the Structure	30

CHAPTER IV RESULTS AND DISCUSSIONS

4.1	Overview	31
4.2	Results for the First Analysis (R1-T08-C4)	32
4.3	Results for the Second Analysis (R2-T08-C14)	37
4.4	Results for the Third Analysis (R3-T08-C6)	42
4.5	High Stress and Displacement at the Middle of Skin (Mountaining effect)	47
4.6	Von Mises Stress and Displacement along the Wing Spar	49
4.7	Results Summary	52