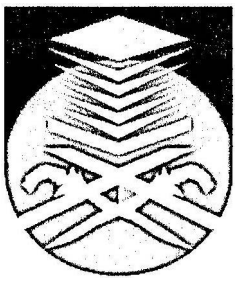

THE COMPUTATIONAL STUDY ON THE AERODYNAMIC CHARACTERISTICS OF AIRFOILS USING FLUENT

ZULKIFLI BIN YAHAYA

(2002334899)

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

Faculty of Mechanical Engineering

Universiti Teknologi MARA (UiTM)

DECEMBER 2005

ACKNOWLEDGEMENT

First and foremost, Alhamdulillah and thanks to Allah The Almighty for the completion of this Final Project on The Computational Study on Aerodynamics Characteristics of Airfoils Using Fluent, as part of the requirements for the award of Degree in Engineering (Hons) (Mechanical).

A variety of people have contributed to the work in this study. Miss Zurriati Mohd Ali (UiTM Penang), my Project Advisor, showed great patience in helping me in understanding of the FLUENT. The help of Mr. Azli Abd Razak (UiTM Shah Alam), as my Co-Advisor, was invaluable. He gave great support at the time most needed. I would like to thank them for always helping and supporting me. Without them, none of this would have been possible.

A special thank to Dean of Mechanical Engineering Faculty, UiTM Shah Alam, Associate Professor Ir. Dr. Abdul Rahman Omar for being very supportive and helpful.

I would also like to thank all the staffs at Universiti Teknologi MARA (UiTM) Shah Alam, especially the staffs at the Faculty of Mechanical Engineering for their very helpful guidance.

My thanks extend to all my friends, for all the helps and supports. For those people who help directly or indirectly in completion of this project, I would like to sincerely thank them for the help.

ABSTRACT

Steady-state, two-dimensional CFD calculations were made for the symmetry airfoil, NACA 0012 and cambered airfoil, WORTMANN FX 63-137 using the standard turbulence model, κ-ε model of commercial code, FLUENT. Comparisons of aerodynamics characteristics, pressure contours, and velocity vectors were made between the two airfoils. The effects of Reynolds numbers, angles of attack, and airfoil profiles on the airfoils were investigated. Results showed that for each and increased Reynolds number with corresponding angle of attack, the cambered airfoil has increasing and higher values of C_L and C_D than the symmetry airfoil. As the Reynolds number and the angle of attack are increased, the stagnation points of both airfoils are moved further backward to the lower surface. The cambered airfoil, WORTMANN FX 63-137, has clear pressure difference between the upper and the lower surfaces. The lower surfaces have higher pressure than upper surface. The highest pressure is at the leading edge, followed by the pressure at trailing edge. For symmetry airfoil, NACA 0012, the upper surface has the lowest pressure. As the Reynolds number and angle of attack are increased, the pressure difference between the upper and lower surfaces is increased. The highest pressure is at the leading edge. Upper surface of NACA 0012 airfoil has the highest velocity vectors. The leading edge has the lowest velocity vectors. For WORTMANN FX 63-137, the slowest velocity is at the leading and trailing edges. The lower surface has lower velocity vectors than the upper surface.

TABLE OF CONTENTS

CHAPTER 1

INTRODUCTION

The flow of air over the surface of an airplane is the basic source of the lifting or sustaining force that allows an airplane to fly. The science that deals with the flow of air or any gas is called aerodynamics. The study of the flow of gases is important to predict the aerodynamic characteristics over a body. The aerodynamics characteristics obtained from the computational simulations will help in designing in the early stage before a body is undergoes the wind tunnel testing and prototyping for real flight simulation

This project is introductory phase of using Computational Fluid Dynamics (CFD) package to predict the aerodynamic characteristics. FLUENT 6 is used as the solver to predict the aerodynamic characteristics over the symmetry airfoil and cambered airfoil, namely NACA 0012 and WORTMANN FX 63-137 respectively. In order to run the simulations, extensive learning on the basic and important parameters of FLUENT 6 is required.

The collected data from the study is useful to validate the data from the wind tunnel testing in the future.