A GUI-BASED NUMERICAL FLOW SOLVER USING THE MACCORMACK'S SCHEME FOR PRANDTL-MEYER EXPANSION WAVE

MUHAMMAD IMRAN BIN BASSERI
(2013692004)

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

Faculty of Mechanical Engineering
University Teknologi MARA (UiTM)

JULY 2017

ACKNOWLEDGEMENT

Foremost, I would like to express my sincere gratitude to my advisor Dr.. Ong Jiunn Chit for the continuous support of my Final Year Project study and 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 Final Year Project study.

Besides my advisor, I would like to thank the rest of my thesis committee: Dr. Lim Teong Yeong, Mr. Fairosidi and Mr. Arif, for their encouragement, insightful comments, and hard questions.

Last but not the least, I would like to thank my family: my parents Basseri Ismail and Nooraini Abu Samah, for giving birth to me at the first place and supporting me spiritually throughout my life.

ABSTRACT

Prandtl-Meyer expansion fan is a flow phenomenon that occurs when a supersonic gas flows past through a convex corner, resulting in the formation of an expansion wave that consists of many Mach waves projected right at the corner. It is an inviscid, two-dimensional flow type that intended to be solved with the means of numerical methods in preference than its exact analytical solutions counterpart, thereby making it fall under the Computational Fluid Dynamics (CFD) subject. The flow solver for the governing Euler equations in strong conservation form will be based on the MacCormack's predictor-corrector explicit finite difference method. The coding for the solution of the flow has been done with the aid of a personal computer using MATLAB r2015a version followed by validation procedures to ensure the agreement of the obtained results with the analytical solution results. Subsequently, GUI-based program was coded and developed by MATLAB Graphical User Interface Development Environment (GUIDE) to make the solver program easier to use since a well-designed graphical user interface can free the user from learning complex command languages to solve the flow problem themselves. The program's main purpose is to be intuitive and serves as education software to assist students to learn and understand the flow phenomena produced by this wave in the desired flow field. User will be required to fill the inlet air properties and the solver will return two types of results, a Mach contour mesh grid representation and the air properties at any x-location in tabulated form.

TABLE OF CONTENTS

CHAPTER 1

INTRODUCTION

1.1 Background

Fluid flows are a common scenario in this world. One can even observe and feel the result of fluid flows in the daily life such as the changes of weather like rain or hurricane, and even the process of human breathing in and out involves the mechanism fluid flows. Now these flows are governed by three conservation laws principles which are conservation of mass, momentum and energy. These principles could then be represented mathematically by a set of partial differential equations (PDE). Then the purpose of Computational Fluid Dynamics (CFD) is introduced, that is to replace the partial differential equations of fluid flow in the flow field with set of numbers with further advancement in terms of time or space which leads to a complete final numerical solution of the designated flow field (Wendt, 2009).

Before computing era started, when encountering a problem, analytical method and empirical methods are used by scientists to derive the solution. Analytical solutions based on the theorems to cast formulas while empirical method derived from experiment and observation rather than theory. These solutions are ultimately accurate and provide hindsight of what was happening. However, analytical solutions can only be derived in limited situation only such as simple geometry and linear models. This result in rather significant amount of energy and time consumed during the