COMPUTATIONAL SIMULATION OF PETROL BEHAVIOR (SWIRL) INSIDE COMBUSTION CHAMBER DURING THE INTAKE-STROKE USING FLUENT

SHAFIE BIN OMAR
(2002333821)

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)

5 DECEMBER 2005

TABLE OF CONTENTS **PAGE**

ACKNOWLEDGEMENT

In the name of Allah, The Most Beneficent and Merciful

I would like to express my sincere gratitude and appreciation to my project advisor, Mr. Azli Bin Abd Razak for his continued support, generous guidance, help, patience and encouragement in the duration for the preparation of this thesis until its completion.

I would also like to take this opportunity to express my thanks to all the lecturers and technician in the Faculty of Mechanical Engineering and CADEM laboratory, especially to Mr. Razip for his instructions on the use of FLUENT. I would like to take this opportunity to appreciate the important contributions of the following persons; Mr. Mohd Ridhwan and Mr Hazleen kamis for allowing me to use the facilities in order to completing this thesis. Beside that, I would like to express my thanks to Mr Ahmad Tarmizi Ahmad Basri and Mr Amir Hamzah Jaafar for their contributions of given the data and some useful information from their thesis.

ABSTRACT

This thesis is presenting the development of computational simulation of petrol behavior inside combustion chamber during the intake-stroke by using fluent. The validation has been conducted using experimental analysis. The data that had been simulated are collected from the experiment done in laboratory. The validation of the result from simulation had been done by comparing with theoretical that stated in the book. The results analyses are base on the difference speed of engine. The swirl phenomenon is proportional to the velocity of fuel that applies to the intake manifold. The higher the velocity of fuel flow into the combustion chamber the more swirl phenomena occur inside combustion chamber.

CHAPTER I

PRELIMINARY WORKS

1.0 Introduction

Automobiles afford such a convenient means of transportation that they will continue to be demanded by our mobile society. As a result, the requirement to meet the challenge of producing cleaner and more efficient power plants will intensify further over the next few years. This challenge requires an increased commitment to research by the transportation industry. The industry has already improved engine performance significantly through the use of new technologies such as ultra-high injection pressure fuel sprays (e.g. to reduce pollutant emission levels) and the use of advanced materials (e.g. Ceramics to influence engine heat transfer losses). More recently, advanced computer models are finding increased use in the industry as a tool to increase the pace of change.

The internal combustion engine represents one of the more challenging fluid mechanics problems to model because the flow is compressible with large density variations, low Mach number (typically < 0.4), turbulent, unsteady, cyclic, and non-stationary, both spatially and temporally [1]. The combustion characteristics are greatly influenced by the details of the fuel preparation process and the distribution of fuel in the engine which is, in turn, controlled by the in-cylinder fluid mechanics.