CHAPTER 1

INTRODUCTION

1.1 INTRODUCTION

The title of this project is Numerical Analysis of In-Cylinder Flow. In this project, the modelling of combustion chamber is needed to determine the flow in the combustion chamber. Combustion chamber has an inlet and outlet, where there is a represent the intake and exhausts port, place to the valve lift, bore and stroke to place the piston and the wall to avoid energy and heat losses to the environment with high rate. The combustion chamber is divided in to two types, one is the combustion chamber for high torque vehicle; in which stroke size is bigger than the bore size and the second types is the high performance vehicle; where the size of bore is bigger than the size of stroke.

In the combustion process, maximum temperature, pressure and the flow of air can analyze. The maximum of temperature, pressure and type of flow in the combustion chamber depend to the velocity and the properties of air that goes through the inlet manifold. The types of flow are divided to two categories, one is internal flow and external flow.

The definition of internal flow is, a flow which confined by a surface. In this flow, the boundary layer is unable to develop without eventually being constrained. External flow is the flow where the boundary layer develops freely without constraint imposed by an adjacent surface.
In this project, the important calculation in computational fluid dynamics is used like in boundary conditions, Reynolds Averaged Navier-Stokes equations and the other calculations. It is important to determine the boundary layer flow, whether there is a laminar or turbulent flow in the combustion chamber. The pattern of flow is also dependent to the valve lift. The increasing flow will give different pattern of flow at the valve lift area.

This study was carried out to obtain the suitable mesh for the maximum static pressure, velocity for magnitude and vector. The meshing process is done to provide the data.

1.2 PROBLEM STATEMENT

Usually to calculate a data when a data is selected from a certain range, the best range is selected to get the approximately best data. To get the approximately best data, the formulation like Runge-Kutta method, Jacobi’s method and the other method in numerical analysis is used.

In this project, the calculation data is not needed, as it provided by the Computational Fluid Dynamics (CFD) software. The total number of iteration and the size of meshing will affect to get the approximately best data. In this CFD software, the basic equations of fluid dynamics is consider in part of to get the result.

1.3 OBJECTIVE OF PROJECT

The objective of this project is to do a numerical analysis of in-cylinder flow and to simulate the pattern of flow in combustion chamber dependent to valve lift (camshafts angle, \(\omega t\); \(\omega\)= crankshaft angular velocity, \(t\)= time)
1.4 SCOPES OF PROJECT

There are four scopes in this project:

(i) The CFD software to modelling the combustion chamber is a STAR CD.

(ii) The model specification for the inlet valve lift in mm is 1.78, 3.55, 5.33, 7.11, 8.89, 10.66

(iii) The model specification for the exhaust valve lift in mm is 1.45, 2.90, 4.36, 5.81, 7.26, 8.71

(iv) Type of flow used in the combustion chamber is turbulence

1.5 PROJECT ORGANIZATIONS

Chapter 2: Find the literature review about turbulence and flow in the cylinder, the flow pattern through the inlet valve, the valve actuation, the basic formula or equations of fluid dynamics, and the boundary layer flow and the CFD tool.

Chapter 3: First work is this chapter is modelling the original combustion chamber using PROSTAR. Then, simulate the flow inside the combustion chamber using PROSTAR. In this chapter, input and output data and the best size of mesh selected were defined.

Chapter 4: Provide detailed information on results, like graph or diagram to represent the result and discussion on this project, which are the prediction of internal flow and the properties of this flow.

Chapter 5: Summaries of result and provide conclusions of the present work. Recommendations for further work is also presented in this chapter to improve the result for future use.