VARIABLE INLET VALVE OPENING AND ITS TURBULENCE SIMULATION USING COMPUTATIONAL FLUID DYNAMICS

[1]M S. AL-KHALDI, [2]M.K.A. ARIFFIN, [3]S.SULAIMAN, [4]B.T.H. BAHRUDIN, [5]N.A. AZIZ

[1,2,3,4,5]Department of Mechanical and Manufacturing Engineering, University Putra Malaysia, Malaysia
E-mail: [1]msjabri1@yahoo.com, [2]khairol@upm.edu.my, [3]shamsuddin@upm.edu.my, [4]hangtuah@upm.edu.my, [5]nuraini@upm.edu.my

Abstract- A new design of inlet valve has been innovated based on a study of flow formation using CFD (Computational Fluid Dynamics) simulation software of two intake valve's designs. Three-dimensional figs of simulations represent flow were used to visual the turbulence kinetic energy of the air that is formed in the combustion chamber. The new design of inlet valve contains holes in the chamfer to allow more swirl turbulence flow than conventional design in order to leverage the good Air-Fuel mixing in the combustion chamber. Simulations carried by two port flow speeds 5 & 15 m/s for different designs and compared with conventional design simulation. Figs showed 60 % improvement in turbulence when using valve contains sided circular holes in simulation. A new Variable Inlet Valve Opening (VIVO) design was proposed based on the promising result in forming swirl and tumble turbulence that should improve burning, torque and power of the engine. The innovated valve design with self drive variable opening holes also should give more breathe during the high speed of the engine.

Index Terms- Inlet valve Optimization, Variable Valve Timing, Fuel efficiency, combustion engine turbulence.

I. INTRODUCTION

Highlight Valves are usually controlling flow in automotive internal combustion engines. Poppet valve is the most common type of valve used which contains of a disc-shaped head having a stem at one side extended to a certain length. There is an edge chamfered from the side of the valve head and it is accurately ground at an angle usually 45 degrees, and sometimes 30 degrees, to form the seating face[1]
Optimization of airflow and fuel spray is very important for the interaction within the cylinder to achieve stable combustion with least emissions. [2] The different flows can be optimized by the following variables;

1. Inlet port design
2. combustion design
3. valve design
4. piston head design (bowl in it)
5. Valve alignment and number of intake valve
6. Shrouding direction of valves

The turbulent air motion in spark ignition engines controls combustion efficiency and formation [3]
In addition to the normal desired turbulence, a rotational motion called swirl is generated on the Air-Fuel mixture during intake valve opening. [4]
Swirl and tumble are type of turbulence and they are considered important factors to improve the reaction of combustion. Swirl is the rotational component whose axis is parallel to the axis of the cylinder but tumble is denoted by the component whose axis is perpendicular to the axis of the cylinder. [5]
The swirl coefficient (Cs), defined as a ratio of the flow angular to axial momentum at each valve lift, can be defined as

$$C_s = \frac{8G}{m\Gamma VB}$$

Where G is impulse mater torque, $m\Gamma$ is the air mass flow rate that enters the port, and B is the cylinder bore. [6]
Swirl usually enhances the mixing of air and fuel to give a homogeneous mixture faster than those mixing have less swirl flow. Swirl also has an effect to have rapid spreading of the flame during the combustion process. [8]
The air motion in cylinder in diesel engines is generally characterized by swirl, squish and turbulence, which have a major impact on Air-Fuel mixing and combustion. Swirl motion of the air is usually controlled by the design of the intake port. The good intake port design will generate higher swirl and help to improve combustion [7]
Murali and Mallikarjuna in their research suggested that the intake airflow analysis would be very much useful for the optimization of the combustion chamber of the modern internal combustion engines.[9]
In general, the purpose of improve and study the flow in combustion chambers is to amplify the combustion rate and to extend the flammability limit, along with improving efficiency, and fast burning. This will reduce hydrocarbon (HC) and carbon monoxide (CO) emissions because of god mixing and better burning.[5]
It is important to have turbulence to speed evaporation of the fuel, to enhance air-fuel mixing, and to increase combustion speed and efficiency. [4] The swirl levels can be measured by CFD and predictions with the same or variable intake port geometries[6]
The flow effect also noticed in compressed Natural Gas (CNG) engines to improving the mixture flow and the flame propagation speed in cylinder. The stronger turbulence intensity leaded to a lean combustion, higher flame propagation speed, and improve the turbulent kinetic energy (TKE) in cylinder, and which should increase the flame propagation. CFD

simulations were successful to analyze the mixture formation in combustion chambers. [10]

II. MATH DESIGN AND SIMULATION

If The most commonly used valve is the poppet valve. It has several advantages over rotary and disc valves [1]. It gives good lubrication, good flow properties, good heat transfer to the cylinder head good seating, and it is cheap in manufacturing. Inlet valves are usually constructed using the one piece design, whereas exhaust valves are normally manufactured using a two piece design.[1]

We have used in our simulation a selected design of combustion chamber as per the fig (1) with matched inlet valve design that has smaller stem than exhaust valve.

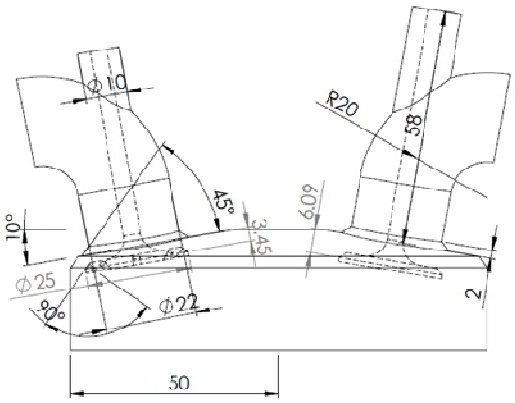


Fig 1: Combustion Chamber (mm)

Analysis system of fluid flow (CFX) in ANSYS software has been used to simulate designed geometries along with designed valves. Then designs were meshed according to parameters in fig (2).

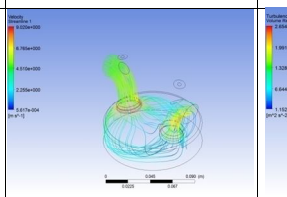
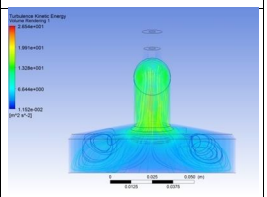
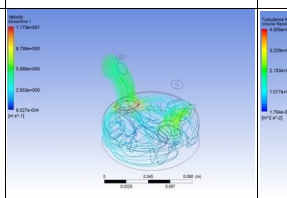
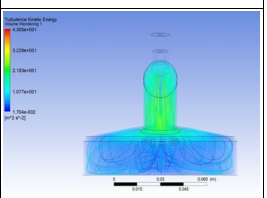
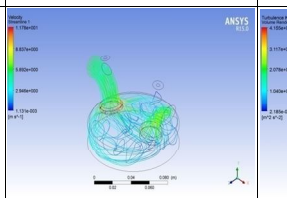
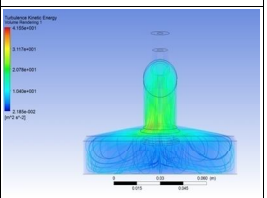
Defaults	
Physics Preference	CFD
Solver Preference	CFX
Relevance	0
Sizing	
Use Advanced Size Fun...	On: Curvature
Relevance Center	Medium
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Slow
Span Angle Center	Fine
Curvature Normal A...	Default (18.0 °)
Min Size	Default (4.4051e-002 mm)
Max Face Size	Default (4.40510 mm)
Max Size	Default (8.81020 mm)
Growth Rate	Default (1.20)
Minimum Edge Length	4.84650 mm
Inflation	
Use Automatic Inflation	Program Controlled
Inflation Option	Smooth Transition
Transition Ratio	0.77
Maximum Layers	5
Growth Rate	1.2
Inflation Algorithm	Pre
View Advanced Options	No
Patch Conforming Options	

Fig 2:Parameters of the meshing

Each valve design had two setups for speed 5 and 15m/s under the boundary details of the inlet valve which starts from the port surface to indicate the start of flow of air. It can be also simulated for air-fuel mixture but material should be selected in the domain at the basic settings. In our experiment we have selected air at 25 C. and boundary details of the exhaust to a relative pressure of 1 (Pa). After that we have run the solutions and got the results of streamlines of air flow by geometry of 50 points. We came out with six different three dimensional air flow figs, representing three different designs with distinguished velocity and turbulence patterns. See table (1).

Multipliers can be especially confusing. Write "Magnetization (kA/m)" or "Magnetization (10^3 A/m)." Do not write "Magnetization (A/m) × 1000" because the reader would not know whether the top axis label in Fig. 1 meant 16000 A/m or 0.016 A/m. Fig labels should be legible, approximately 8 to 12 point type.

Table 1: comparison of velocity and turbulence between different designs

	Flow speed inlet port 5 m/s Streamline-Isometric view (Y up)	Flow speed inlet port 15 m/s Volume rendering-View from +X
Inlet Valve with no holes conventional design	 *Stream line beteen (.0005-9.02) m/s* *Turbulence Kinetic Energy (.004-3.96)m^2/s^2*	 *Stream line between (.002-27.39) m/s* *Turbulence Kinetic Energy (.015-26.54)m^2/s^2*
3Inlet Valve with 8 holes dia .5 mm - circular holes	 *Stream line between (.0008-11.73) m/s Turbulence Kinetic Energy (.006-7.63)m^2/s^2*	 *Stream line between (.002-35.26) m/s* *Turbulence Kinetic Energy (.017-43.05)m^2/s^2*
Inlet Valve with 8 holes dia .5 mmX 1 mm - oval holes	 *Stream line between (.001-11.78) m/s* *Turbulence Kinetic Energy(.007-6.99)m^2/s^2*	 *Stream line between (.006103-36.97) m/s* *Turbulence Kinetic Energy (.022-41.55)m^2/s^2*

III. ANALYSIS OF THE RESULTS

The calculation has been taken under air flow inlet 5 m/s and 15 m/s that enter the port of inlet valve with 1 Pascal pressure in outlet valve. The simulations of flow in combustion chamber, and information about swirl in cylinder have been obtained as 3D figs with 60 lines of air direction flows. We have taken two design of holes one of them rounded shape with 1 mm diameter and with direction 45 degree from the surface

of the valve head toward cylinder surface. The other design has oval shape and open through the valve head with also 45 degree from the piston surface. In each design there are 8 holes circulating the valve stem with similar distances. Both designs and also the conventional design have been simulated to get the flow in the cylinder at two flow speeds. In the table (1) we can see the 3D camera view that shows conditions of flow in the three designs. Swirls and turbulences can be obtained from the fig of velocity fields along with streamline pattern and Turbulence Kinetic Energy (TKE) volume rendering. We find that best turbulence from the simulation within the valve design which has circular holes sizing (0.5) mm, and also we find the higher velocity obtained from the simulation of the valve design that have oval shaped hole sizing (0.5X1 mm).

For valve design contains circular holes, we got TKE (.017-43.05)m2/s2 , whereas standard valve design with no holes gave result of TKE (.015-26.54)m2/s2 .But for valve contains oval shaped holes, the result of TKE was (.022-41.55)m2/s2 which is giving little less turbulence than circular holes, therefore oval holes resulted more (TKE) speed in the design. The result was shows generation of more swirls and tumbles from the design has (.5 mm) circular holes than conventional valve design. But the flow in the design has oval shaped holes were having more lean flows. More turbulence leads better combustion efficiency than conventional design and more opening through valve gives more breathe.

IV. NEW VARIABLE INLET VALVE OPENING (VIVO) DESIGN

Since there are relatively few studies available that document the air flow for valve designs and hence it is important to control the turbulence and mixing of the fuel, we came up with new design that can be self controlled to increase the turbulence in high speed as well as open more room for the breathing in a high speed gradually. Normally, the air pressure is increased proportionally to the rotation speed of crank shaft. We have utilized this phenomenon to create a rotating disk on the head of the intake valve with carving in the head of the holes of the disk. The head can be attached to the valve head by thermal affect or cooling the head part using liquid nitrogen then attaching it to the base of valve head. The disk part has circular grove parallel for surface head that match a cavity in the centre of the valve head. After adding suitable spring around the grove that is perpendicular to the surface disk, we will be able to get a rotating disk to certain angle when applying a pressure around the stem valve axis. With the help of cavities, gaps in between matched two parts are opening gradually when applying pressure until reaching the maximum opening. The timing of opening and strength of pressure required for moving the desk can be controlled by the spring. See fig (3).

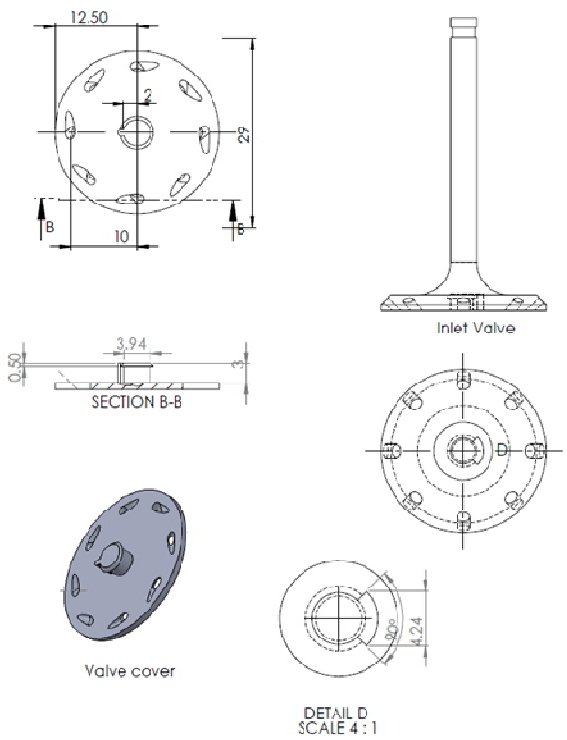


Fig 3: Upper part and lower part of the new design

CONCLUSION

Two designs of inlet valves with sided holes were created in the conventional lollipop valve for specific geometry. One design contains circular holes and the other with oval shaped holes, and both are perpendicular to the side chamfer of the valve head toward the direction of the centre of the combustion chamber. The designs along with conventional inlet design were simulated by ANSYS software in CFX tools box option using an estimated combustion chamber. The simulations were done for three similar geometries; represent the conventional inlet valve design and represent the new two designs. The simulation carried out for two speeds 5 & 15 m/s. Velocity steam line of valve design contains circular holes was increased up to 92% that the conventional valve design with no holes and also the turbulence increased by 62%. The valve design of oval holes showed little higher velocity streamline than valve design contains circular holes, but it showed 3.5% less in turbulence.

A new design (VIVO) was proposed to have a self driven side holes opening by the affect of air flow pressure. It was found the variable opening not only gives better turbulence but also gives more breathe to the combustion chamber in the high speed of engine. The new design is in primary stage and can be further modified to design the spring in the core of the valve head to match the speed of the engine. The diagonal side opening in the valve head can be increased or decreased at specific timing according to the need in the combustion engine by controlling the strength of the spring, holes' dimension and holes' cavities.

REFERENCES

[1] Lewis, R. and R.S. Dwyer-Joyce, Automotive engine valve recession. 2002: Professional Engineering.

[2] Moon, S., et al., The influence of airflow on fuel spray characteristics from a slit injector. Fuel, 2007. 86(3): p. 400-409.

[3] Harshavardhan, B. and J. Mallikarjuna, Effect of piston shape on in-cylinder flows and air–fuel interaction in a direct injection spark ignition engine–A CFD analysis. Energy, 2015.

[4] Pulkrabek, W.W., Engineering fundamentals of the internal combustion engine. Vol. 478. 1997: Prentice Hall Upper Saddle River, NJ.

[5] Hill, P. and D. Zhang, The effects of swirl and tumble on combustion in spark-ignition engines. Progress in energy and combustion science, 1994. 20(5): p. 373-429.

[6] Kang, K.Y. and R.D. Reitz, Intake flow structure and swirl generation in a four-valve heavy-duty diesel engine. Journal of engineering for gas turbines and power, 2000. 122(4): p. 570-578.

[7] Prasad, B., et al., High swirl-inducing piston bowls in small diesel engines for emission reduction. Applied Energy, 2011. 88(7): p. 2355-2367.

[8] Taylor, C.F., The Internal Combustion Engine in Theory and Practice, Volume 2: Combustion, Fuels. Materials, Design, 1985.

[9] Murali, K. and J. Mallikarjuna. Effect of Engine Speed on Intake Valve Flow Characteristics of a Diesel Engine An Analysis Using Particle Image Velocimetry. in Emerging Trends in Engineering and Technology, 2008. ICETET'08. First International Conference on. 2008. IEEE.

[10] Wu, C., K. Deng, and Z. Wang, The effect of combustion chamber shape on cylinder flow and lean combustion process in a large bore spark-ignition CNG engine. Journal of the Energy Institute, 2015.

★★★