Global Journal of Researches in Engineering
Mechanical and Mechanics Engineering
Volume 12 Issue 5 Version 1.0 Year 2012
Type: Double Blind Peer Reviewed International Research Journal
Publisher: Global Journals Inc. (USA)
Online ISSN: 2249-4596 Print ISSN:0975-5861

CFD Analysis of Intake Valve for Port Petrol Injection SI Engine

By K.M Pandey & Bidesh Roy
NIT Silchar, India.

Abstract - The air standard efficiency for SI engine is approximately 60% under full load condition but the actual brake thermal efficiency under full load condition is approximately 32.6% which is due to the various losses that occur. One of the primary lose is burning time loss which is approximately 4% and occur due to finite time combustion of the charge. This lose can be reduced to some extend by generation of a higher degree of swirl which will increase turbulence intensity with in the engine cylinder. The production of turbulence of higher intensity is one of the most important factors for stabilizing the ignition process, fast propagation of flame, especially in case of lean-burn combustion In general, two type of vortices are utilized in order to generated and preserve the turbulence flows efficiently. These vortices are usually known as swirl and tumble flows, which are organized rotations in the horizontal and vertical plane of the engine cylinder, respectively.

Keywords : Swirl, turbulence intensity, swirl ratio.

GJRE-A Classification : FOR Code : 091399

CFD ANALYSIS OF INTAKE VALVE FOR PORT PETROL INJECTION SI ENGINE

Strictly as per the compliance and regulations of:

Open Association of Research Society, USA
Research | Diversity | Ethics

CFD Analysis of Intake Valve for Port Petrol Injection SI Engine

K.M Pandey [α] & Bidesh Roy [σ]

Abstract - The air standard efficiency for SI engine is approximately 60% under full load condition but the actual brake thermal efficiency under full load condition is approximately 32.6% which is due to the various losses that occur. One of the primary lose is burning time loss which is approximately 4% and occur due to finite time combustion of the charge. This lose can be reduced to some extend by generation of a higher degree of swirl which will increase turbulence intensity with in the engine cylinder. The production of turbulence of higher intensity is one of the most important factors for stabilizing the ignition process, fast propagation of flame, especially in case of lean-burn combustion In general, two type of vortices are utilized in order to generated and preserve the turbulence flows efficiently. These vortices are usually known as swirl and tumble flows, which are organized rotations in the horizontal and vertical plane of the engine cylinder, respectively. They contribute to the improvement of engine performance. Hence, it is indispensable for the development of an ICE with high compression ratio to realize high turbulence intensity and lean burn combustion. Swirl can be generated during intake stroke as well as compression stroke of the engine. Intake generated swirl usually persists through the compression, combustion, and expansion stroke and it can greatly enhances the mixing of air and fuel to give a homogeneous mixture in the very short time. This is done by shaping and contouring the intake manifold, valve ports, and by use of shrouded intake valve.

Keeping the above point in view, in this paper, an analysis is performed in a port fuel injection SI engine using computational fluid dynamic (CFD) code FLUENT to determine the level of intake swirl induced by poppet intake valve and its reduction along the length of the cylinder. From the study it was found that intensity of intake swirl reduces along the length of the engine cylinder.

Keyword : *Swirl, turbulence intensity, swirl ratio.*

I. Introduction

The engine cycle of typical internal combustion engines consist of four consecutive processes as intake, compression, expansion (including combustion) and exhaust. Of these four processes, the intake and compression stroke is one of the most important processes which influences the pattern of air flow structure coming inside cylinder during intake stroke and generates the condition needed for the fuel injection during the compression stroke. As a result of the high velocity inside the internal combustion engine (ICE) during operation, all in cylinder flows are typically turbulent. The exception to this is the flows in the corners and small crevices of the combustion chamber where the close distance of the walls diminished out turbulence. Heat transfer, evaporation, mixing and combustion rates all increase as engine speed increases. This increases the time rate of fuel evaporation, the mixing of the fuel vapor and air as well as combustion process. Fluid motion within the engine cylinder is one of the major factors that control the fuel-air mixing and combustion process in spark ignition engines. It also has a significant impact on heat transfer. Both the bulk fluid motion and the turbulence characteristics of the flow are essential to produce the homogeneity structure of air flow come into cylinder. Generally, the initial in-cylinder flow pattern is set up by the intake process and then be substantially modified during compression process. The small-scale mixing of turbulence with compressible flows is represented by the turbulence kinetic energy and turbulence kinematic viscosity .Turbulence inside the cylinder is high during the intake and then decreases as the flow rate slows near bottom dead centre (BDC). It increases again during the compression stroke as swirl, squish and tumble increase near top dead centre (TDC) [1]. Intake generated swirl usually persists through the compression, combustion, and expansion stroke and it can greatly enhances the mixing of air and fuel to give a homogeneous mixture in the very short time. It is also a main mechanism for very rapid spreading of the flame front during the combustion process [2]. Many researchers worked in this area via experimental as well as computational to explorer the phenomenon of the in-cylinder flow of Internal Combustion Engine. Some of them are cited here. B. Reveille and A. Duparchy [3] worked on 3D CFD analysis of an abnormally rapid Combustion phenomenon in downsized gasoline engines. This paper has focused on a particular abnormally rapid, yet non destructive and seemingly stable combustion phenomena which have been identified on low speed mid to high load operating points when performing aggressive downsizings on various engines. Franz X. Tanner & Seshasai Srinivasan [4] worked on CFD-based optimization of fuel injection strategies in a diesel engine using an adaptive gradient method. A gradient-based optimization tool has been developed and, in conjunction with a CFD code, utilized in the search of new optimal fuel injection strategies. The approach taken uses a steepest descent method with an adaptive cost function, where the line search is

Author α, σ : Department of mechanical engineering, NIT Silchar, India.
E-mail : kmpandey2001@yahoo.com

performed with a backtracking algorithm. Vijaya Kumar Cheeda, R. Vinod Kumar, G. Nagarajan [5] worked on design and CFD analysis of a regenerator for a turboshaft helicopter engine. In this paper a continuous heat transfer regenerator for a turboshaft helicopter engine is designed suitably. The regenerator effectiveness is assessed by the CFD tool CFX and evaluated the effectiveness and the pressure drop. The predicted CFD results are in good agreement with experimental results. L. Li, X.F. Peng, and T. Liu [6] worked on combustion and cooling performance in an aero-engine annular combustor. The investigation was conducted to understand the characteristics of the flow, combustion, cooling performance and their interaction in an aero-engine combustor. The conservation equations and Eddy-dissipation combustion model were employed for solving the flow, heat transfer, and combustion in the entire combustor. The reliability of the simulation was demonstrated by comparing calculated combustor exit temperature distributions with profiles of the rig-test measurements. Christian Hasse, Volker Sohm, and Bodo Durst [7] worked on Numerical investigation of cyclic variations in gasoline engines using a hybrid URANS/LES modeling approach. The study investigates the feasibility of using the SST DES model to predict cycle to cycle variations in internal combustion engines and the effect of cyclic variations in engines and their root causes including the major flow patterns. Wendy Hardyono Kumiawan, Shahrir Abdullah and Azhari Shamsudeen [8] worked on CFD study of cold-flow analysis for mixture preparation in a motored four-stroke direct injection engine. In this study, the CFD simulation to investigate the effect of piston crown to the fluid flow field inside the combustion chamber of a four-stroke direct injection automobile engine under the motoring condition is presented. The analysis is focused on study of the effect of the piston shape to the fluid flow characteristics the result obtained from the analysis could be employed to examine the homogeneity of air-fuel mixture structure for better combustion process and engine performance. Andras Kadocsa, Reinhard Tatschl and Gergely Kristof [9] worked on analysis of spray evolution in internal combustion engines using numerical simulation. This paper summarizes results of research about a new approach of spray formation calculations. Using a primary breakup model for separately describing the initial liquid disintegration of injected liquid based on the flow properties stemming from a previous calculation of injector nozzle flow gives a better prediction capability and suits the new needs of advanced combustion systems such as HCCI engines or various forms of split injection. Toyoshige Shibata Hideo Matsui, Masao Tsubouchi and Minoru Katsurada [10] worked on Evaluation of CFD Tools Applied to Engine Coolant Flow Analysis. This paper presents the results of test application of some automatic mesh generation tools to the CFD calculation of coolant flow, and compares the functional characteristics and features of these tools. The paper also discusses coolant flow items that can be evaluated by CFD analysis and the merits of applying CFD to these items. Semin, N.M.I.N. Ibrahim, Rosli A. Bakar and Abdul R. Ismail [11] worked on In-Cylinder Flow through Piston-Port Engines Modeling using Dynamic Mesh. This paper presents numerical study of three-dimensional analysis of two-stroke spark-ignition cross loop-scavenged port. The objective of this study is to investigate the in-cylinder characteristics at motored transient condition. The pressure on in-cylinder and intake port were collected and applied for validation with numerical results for 1400 rpm. The three-dimensional modeling analysis was performed utilizing dynamic mesh method. The prediction of distribution of in-cylinder pressure and mass fraction of gases function of crank angle were discussed. The results shown that the relative error between experimental and numerical less that 2 %. Helmut Doleisch [12] worked on simvis: interactive visual analysis of large and time-dependent 3d simulation data. In this paper the major new technological concepts of the SimVis approach are presented and real-world application examples are given. SimVis is a system for the graphical analysis of simulation data, built on a new, cutting-edge technological approach for interactive visual analysis of large, multi-dimensional, and time-dependent data sets resulting from CFD simulation. S. M. Jameel Basha, P. Issac Prasad and K. Rajagopal [13] worked on simulation of in-cylinder processes in a DI diesel engine with various injection timings. In this paper an attempt has been made to study the combustion processes in a compression ignition engine and simulation was done using computational fluid dynamic (CFD) code Fluent. An Axisymmetric turbulent combustion flow with heat transfer is to be modeled for a flat piston 4-stroke diesel engine. The unsteady compressible conservation equations for mass (Continuity), axial and radial momentum, energy, species concentration equations can express the flow field and combustion in axisymmetric engine cylinder. Turbulent flow modeling and combustion modeling was analyzed in formulating and developing a model for combustion process. R. Rezaei, S. Pischinger, P. Adomeit and J. Ewald [14] worked on Evaluation of CI In-Cylinder Flow using optical and numerical techniques. In this paper different port concepts for modern Compression-Ignition engines, usually quantities as the swirl level and the flow coefficient are evaluated, which are measured on a stationary flow test bench. As additional criterion, in this work, the homogeneity of the swirl flow is introduced and defined quantitatively. Different valve lift strategies are evaluated using three-dimensional Particle Imaging Velocimetry in a stationary flow configuration and transient In-Cylinder CFD simulation using both the Reynolds Averaged Navier Stokes equation and the

Global Journal of Researches in Engineering (A) Volume XII Issue V Version I Year 2012

Large Eddy simulation approach. M.M.Noor1, K.Kadirgama1, R.Devarajan, M.R.M.Rejab, N.M.Zuki N.M. and T.F.Yusaf [15] worked on Development of a High Pressure Compressed Natural Gas Mixer for A 1.5 Litre CNG-Diesel Dual Engine. In this paper Computational Fluid Dynamics (CFD) analysis software was used to study the flow behavior of compressed natural gas (CNG) and air in a CNG-air mixer to be introduced through the air inlet of a CNG-Diesel dual fuel stationary engine. Yasar Deger, Burkhard Simperl and Luis P. Jimenez [16] worked on Coupled CFD-FE-Analysis for the Exhaust Manifold of a Diesel Engine. This paper aims to investigate the thermo-mechanical behaviour of an exhaust manifold which has an active cooling system, the full water flow, partial water flow (by 50% reduced cooling flow) and Vapour flow three cases of cooling analyzed. Fluid flow, thermal heat transfer and stress analysis are coupled for each case using a one-way coupling approach. Selected results given in form of temperature, stress and displacement distribution plots in this paper. The investigation was focusing on potential structural optimization measures. Therefore some suggestions for design improvements are presented also, which are presumably effective to reduce the temperature peaks and temperature gradients and to ensure a longer service life for the exhaust manifold. Kihyung Lee, Choongsik Bae, and Kernyong Kang [17] worked on the effects of tumble and swirl flows on flame propagation in a four-valve S.I. engine. The effects of in-cylinder flow patterns, such as tumble and swirl flows, on combustion were experimentally investigated in a four valve S.I. engine. Tumble flows were generated by intake ports with entry angles of 25°, 20° and 15°. Inclined tumble (swirl) flows were induced by two different swirl control valves. The initial flame propagation was visualized by an ICCD camera, the images of which were analyzed to compare the enflamed area and the displacement of initial flames. The combustion duration was also calculated by the heat release analysis. B. Murali Krishna and J. M. Mallikarjuna [18] worked on Tumble flow analysis in an unfired engine using particle image velocimetry. This paper deals with the experimental investigations of the in-cylinder tumble flows in an unfired internal combustion engine with a flat piston at the engine speeds ranging from 400 to 1000 rev/min., and also with the dome and dome-cavity pistons at an engine speed of 1000 rev/min., using particle image velocimetry and It is suggested in the paper to use the flat piston rather than dome, dome-cavity pistons which are rather difficult to manufacture as far as tumble flows are concerned. B. Khalighi worked on Study of the intake tumble motion by flow visualization and PTV [19].The purpose of this work is to characterize the in-cylinder tumbling flow generated by an engine head during the induction process using flow visualization and PTV. The study was carried out for a 4-valve engine head with shrouded intake valves in special single cylinder transient water analog. This shrouded intake valve configuration was used to obtain a prototypical "pure tumble" flow suitable for fundamental combustion studies. K.M Pandey, S.N Pandey, and Bidesh Roy [20] worked on numerical analysis to determine the effect of temperature on the intake generated swirl for port fuel injection SI engine. Hence, for computational investigation for intake swirl within the engine, cold flow simulation will provide faster computational result. In this study it was concluded that the temperature on various part of the engine produces a very negligible effect on the intake swirl generation. Thus, we can see that very few works have been done in field of determining the behavior of intake swirl red along the length of the engine cylinder.

II. Specification of the Si Engine

The engine considered for the computation analysis is a single-cylinder continuous type port fuel injection four stroke SI engine with cylindrical combustion chamber and single intake port and exhaust port. The computation analysis is performed at WOT maximum power condition. The specification of engine is listed in Table 1.

Table 1 : Engine specifications and calculation conditions

Bore x stroke	95mm x 99mm.
Compression ratio	9:1
Piston cavity	Flat.
Max power at WOT	13.2 BHP at 4950 RPM.
Intake valve diameter	42mm
Maximum intake valve lift	12mm.
Exhaust valve opening	64° BBDC.
Exhaust valve closure	5° ATDC.
Intake valve opening	5° BTDC.
Intake valve closure	60° ABDC.
Fuel	C_8h_{18}

III. Poppet Intake Valve

A Poppet intake valve is used in the SI engine in which the computational analysis is performed. The dimensions of the Poppet intake valve are shown in the figure below:

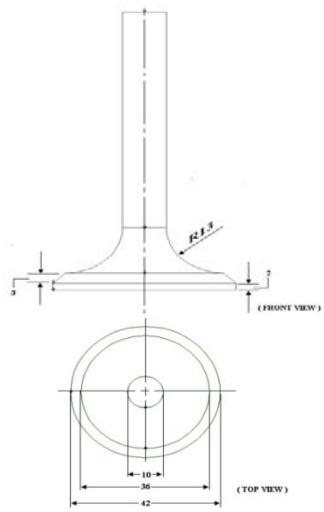

Figure 1 : Dimensions of Poppet intake valve (All dimensions are in mm)

IV. Computational Domain and Boundary Conditions

The numerical formulation of the problem is incomplete without prescribing boundary conditions, which correspond to the specific physical model. The specification of mathematically correct boundary conditions that ensure the uniqueness of the solution, while being compatible with the physics at the boundaries, is not always straightforward. Before arriving at the boundary conditions at various boundaries, we have to first identify the solution/computational domain of the problem. The physical domain and computational domain usually differ. However, the computational domain largely depends on the geometry of physical domain. The computational domain boundary (truncated from the real boundary) along with appropriate boundary conditions should be chosen in such a way that there is negligible change in the results with further increase in its size.

The computational domain shown in the figure 2 is a generalized one since, the analysis is performed at different crank angle during the suction stroke of the engine as result the distance of the piston from the engine head shown in the figure 2 by “B” also varies corresponding to the engine crank angle.

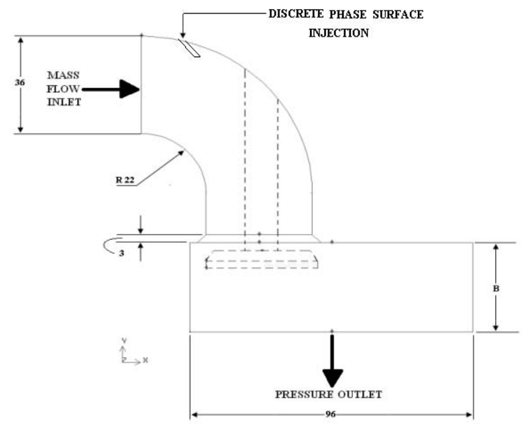

Figure 2 : Computational domain and boundary conditions (All dimension in mm)

The boundary conditions shown in figure 5 are as follows:

I. *Inlet boundary on the inlet port of the engine:* - The inlet boundary condition is assigned as mass flow inlet. Since the investigation is performed at 72 degree of the crank angle and at that instant the mass flow inlet of air is 0.01319 kg/sec for the computation.

II. *Solid surface of the cylinder of the engine:* - It is assigned wall boundary condition i.e. no slip condition on the solid surface of the cylinder. The computation is performed with solid surface of the cylinder at a temperature of 300°K for faster computational result [20].

III. *Outlet Boundary on the piston of the engine:* - Outlet boundary is assigned the pressure outlet boundary condition. For the investigation outlet pressure is taken as a static pressure of 0:935 bar.

IV. *Discrete phase surface injection for injector:* - In the computation domain the injector of the valve is assign as discrete phase surface injection with fuel flow rate of 0.0011 kg/sec for the engine considered.

V. Grid Independence Study

The resolution of the grid has a great quantitative impact over the results obtained. There exists a level of refining of a computational domain beyond which there is no significant quantitative changes in the results achieved. The computational domain at this level of refinement is said to enter the regime of grid independence. In the present work maximum tangential velocity at a surface 9.18mm from engine cylinder head has been taken as the criteria and the number of grid is refined until the required value is gained. For the simulation grid independence was reached for 384876 cells and 82377 nodes as shown in table 2.

Table 2 : Grid independence study

Refining Level	No. of Nodes	No. of cells	Max. Tangential velocity at a surface 9.18mm from engine cylinder head
1)	53307	254668	10 m/sec
2)	52208	244537	10 m/sec
3)	82377	384876	8.8 m/sec

VI. Result and Discussion

Computational result at 72 ° crank angle for the specified SI engine at various locations along the length of the engine cylinder is shown below:-

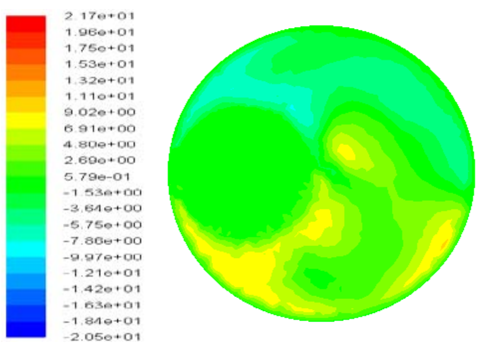

Figure 3 : Contour plot of tangential velocity (m/sec) for surface located at 9.18mm from Engine cylinder head.

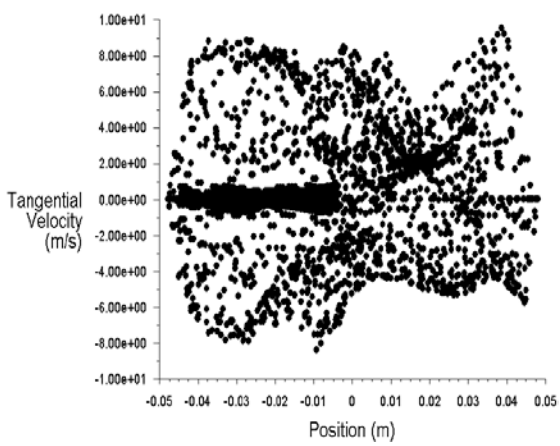

Figure 4 : X-Y plot of tangential velocity (m/sec) for surface located at 9.18mm mm from Engine cylinder head.

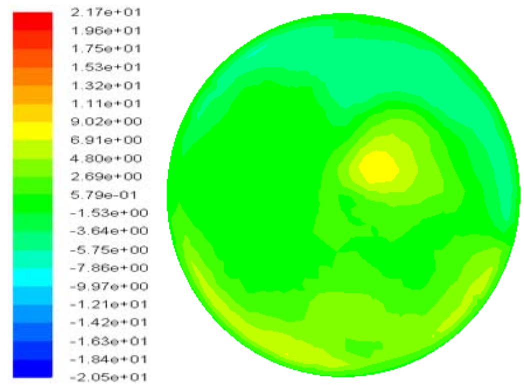

Figure 5 : Contour plot of tangential velocity (m/sec) for surface located at 18.1mm mm from Engine cylinder head.

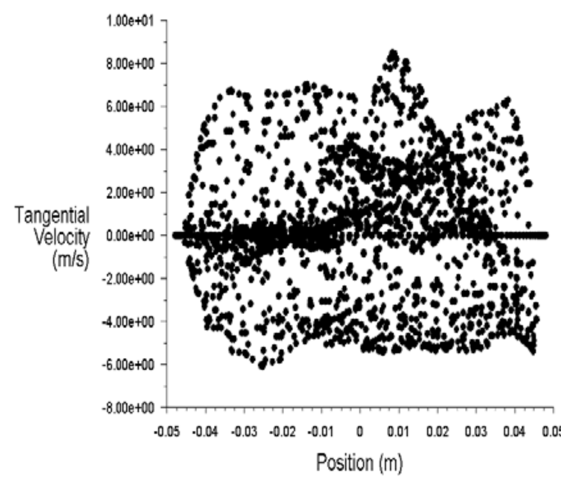

Figure 6 : X-Y plot of tangential velocity (m/sec) for surface located at 18.1mm mm from Engine cylinder head.

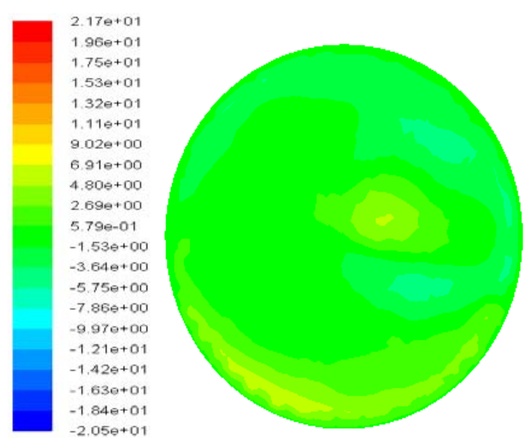

Figure 7 : Contour plot of tangential velocity (m/sec) for surface located at 28.8mm mm from Engine cylinder head.

Global Journal of Researches in Engineering (A) Volume XII Issue V Version I Year 2012

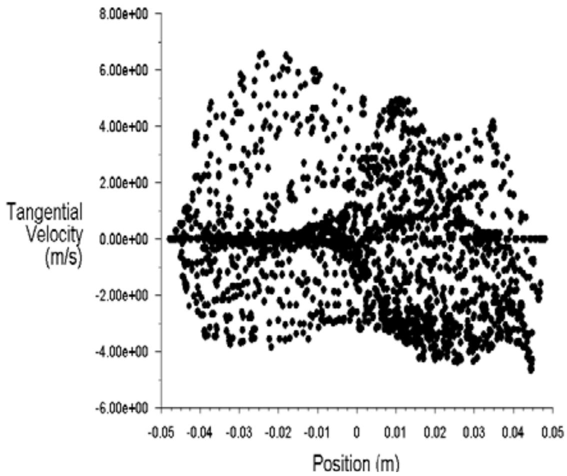

Figure 8 : X-Y plot of tangential velocity (m/sec) for surface located at 28.8mm mm from Engine cylinder head

The nature of swirling flow in actual operating engine is very difficult to determine. Swirl ratio is a dimensionless parameter used to quantify swirling flow within the cylinder as shown by the equation below

$$SR\ (\Theta) = \left| \frac{v\ (\Theta) x\ 60}{2\Pi nr} \right| \qquad (1)$$

From the equation 1, it is clear that tangential velocity plays a vital role in determining the intensity of swirl within the engine.

From the results of the computation analysis carried out at 72 ° crank angle with poppet intake valve, for the specified SI engine it is seen that the surface at 9.18mm from engine cylinder head which is closer to the valve shows higher tangential velocity at various location compared to the surface at 18.1mm and 28.8mm from engine cylinder head which is at higher distance from the intake valve.

VII. Conclusion

From this study the following it can be concluded that the surface which is closer to the poppet intake valve shows higher tangential velocity at various locations compared to the surfaces which are at higher distance from the intake valve i.e. the intensity of swirl decreases along the stroke length of the engine cylinder.

NOMENCLATURE	
Θ	crank angle (degrees)
r	radial coordinate (m)
v	tangential velocity (m/s)
n	engine speed (rpm)
BTDC	before top dead center
BBDC	before bottom dead center
ATDC	after top dead center
ABDC	after bottom dead center
SR	Swirl ratio

References Références Referencias

1. W.H. Kurniawan; S. Abdullah and A. Shamsudeen, Turbulence and Heat Transfer Analysis of Intake and Compression Stroke in Automotive 4-stroke Direct Injection Engine, Algerian Journal of Applied Fluid Mechanics | Vol. 1 | 2007, pp. 37-50.
2. J.B Heywood, Internal combustion engine fundamental. New York: McGraw-Hill, 1988.
3. B. Reveille and A. Duparchy, 3D CFD analysis of an abnormally rapid Combustion phenomenon in downsized gasoline engines. Oil & Gas Science and Technology – Rev. IFP, Vol. 64 (2009), No. 3, pp. 431-444
4. Franz X. Tanner & Seshasai Srinivasan, CFD-based optimization of fuel injection strategies in a diesel engine using an adaptive gradient method. Applied Mathematical Modelling 33 (2009), pp. 1366–1385.
5. Vijaya Kumar Cheeda, R. Vinod Kumar, G. Nagarajan, Design and CFD analysis of a regenerator for a turbo shaft helicopter engine. Aerospace Science and Technology 12 (2008), pp. 524–534.
6. L. Li, X.F. Peng, and T. Liu, Combustion and cooling performance in an aero-engine annular combustor. Applied Thermal Engineering 26 (2006), pp. 1771–1779.
7. Christian Hasse , Volker Sohm, and Bodo Durst, Numerical investigation of cyclic variations in gasoline engines using a hybrid URANS/LES modeling approach, Computers & Fluids, 2009 (article in press, Contents lists available at Science Direct).
8. Wendy Hardyono Kumiawan, Shahrir Abdullah and Azhari Shamsudeen, CFD study of cold-flow analysis for mixture preparation in a motored four-stroke direct injection engine. Journal of applied science 7(19):2710-2724, 2007.

9. Andras Kadocsa, Reinhard Tatschl and Gergely Kristof, Analysis of spray evolution in internal combustion engines using numerical simulation, Journal of Computational and Applied Mechanics, Vol. 8, No. 1, (2007), pp. 85–100.
10. Toyoshige Shibata Hideo Matsui, Masao Tsubouchi and Minoru Katsurada, Evaluation of CFD Tools Applied to Engine Coolant Flow Analysis, Mitsubihi motors technical review 2004, No.16, pp. 56-60.
11. Semin, N.M.I.N. Ibrahim, Rosli A. Bakar and Abdul R. Ismail, In-Cylinder Flow through Piston-Port Engines Modeling using Dynamic Mesh, Journal of Applied Sciences Research, 4(1): 58-64, 2008.
12. Helmut Doleisch, SIMVIS: Interactive visual analysis of large and time-dependent 3D simulation data, Proceedings of the 2007 Winter Simulation Conference, pp. 712–720.
13. S. M. Jameel Basha, P. Issac Prasad and K. Rajagopal, Simulation of in-cylinder processes in a DI diesel engine with various injection timings, ARPN journal of engineering and applied sciences, vol. 4, no. 1, February 2009, pp. 1-7.
14. R. Rezaei, S. Pischinger, P. Adomeit and J. Ewald, Evaluation of CI In-Cylinder Flow using optical and numerical techniques, SAE ICE conference September 2009.
15. R.Devarajan, M.R.M.Rejab, N.M.Zuki N.M. and T.F.Yusaf, Development of a High Pressure Compressed Natural Gas Mixer for A 1.5 Litre CNG-Diesel Dual Engine. National Conference on Design and Concurrent Engineering, 2009, pp. 435-438.
16. Yasar Deger, Burkhard Simperl and Luis P. Jimenez, Coupled CFD-FE-Analysis for the Exhaust Manifold of a Diesel Engine, 2004 ABAQUS Users' Conference, pp. 1-10.
17. Kihyung Lee, Choongsik Bae and Kernyong Kang, The effects of tumble and swirl flows on flame propagation in a four-valve S.I. engine, Applied Thermal Engineering 27 (2007) pp. 2122–2130.
18. B. Murali Krishna and J. M. Mallikarjuna, Tumble flow analysis in an unfired engine using particle image velocimetry, World Academy of Science, Engineering and Technology 54, 2009, pp. 430-435.
19. B. Khalighi, Study of the intake tumble motion by flow visualization and particle tracking velocimetry, Experiments in Fluids 10, 230-236 (1991).
20. K.M Pandey, S.N Pandey, and Bidesh Roy, Numerical analysis to determine the effect of temperature on the intake generated swirl for port fuel injection SI engine, in proceeding of National conference on Emerging trends in mechanical engineering (ETME-2010), May 14-15,2010.

This page is intentionally left blank