International Journal of Latest Engineering Research and Applications (IJLERA) ISSN: 2455-7137

Volume – 02, Issue – 08, August – 2017, PP – 34-46

CFD Analysis of Single Cylinder IC Engine Inlet Swirl Valve

[1]M D Raj Kamal, [2]S.Kaliappan, [3]S.Socrates, [4]G.Jagadeesh Babu

[1,2,3]Assistant Professor, Department of Mechanical Engineering, Velammal Institute of Technology, Chennai-601204, India

[4]Student, Department of Mechanical Engineering, Velammal Institute of Technology, Chennai, India

Abstract: The intent of this thesis is to design and test a variable swirl and ignition sites. The design aspect used Ansys cfx to model air flow within the cylinder. Swirl rate and volumetric flow rate were calculated from the results. Many design iterations took place before a suitable design was accomplished. Valve guides and seats were installed in the head. The inlet valve is creating an air swirl in the inside of a cylinder. The air motion plays a very important role in fuel-air mixing, combustion and emission formation. Swirl motion of the air is usually generated due to the design of the intake port. A good intake port design will generate higher swirl and help to improve combustion. It helps to reducing pollutant emissions and for better fuel economy. These engines tend to attain the maximum efficiency. CFD enables numerical experiments (i.e. computer simulations) in a 'virtual flow laboratory'. They contribute to the improvement of engine performance.

Key words: CFX Software, Swirl, Valve

1. Introduction

Combustion and fluid flow modeling in an internal combustion engine present one of the most challenging problems. This is due to large density variations where, the fluid motion inside a cylinder is turbulent, unsteady, non-stationary both spatially and temporally. The combustion characteristics were greatly influenced by the details of the fuel preparation and the fuel distribution in the engine cylinder, which was mainly controlled by the in-cylinder fluid dynamics.

Pollutant emissions were controlled by the turbulent fuel–air mixing and combustion processes. A detailed understanding of these processes is required to improve performance and reduce emissions without compromising the fuel economy. Several types of research performed experimental and numerical studies towards the parameter optimization, fluid mechanics, and combustion phenomena.

The effects of combustion chamber geometry on combustion process in an optically accessible. Flame movement behaviors such as the distribution of flame velocity vectors and the averaged flame velocity inside and outside the combustion chamber are measured by means of cross-correlation method. Studies indicate that re-entrant combustion chamber is better than dish chamber.

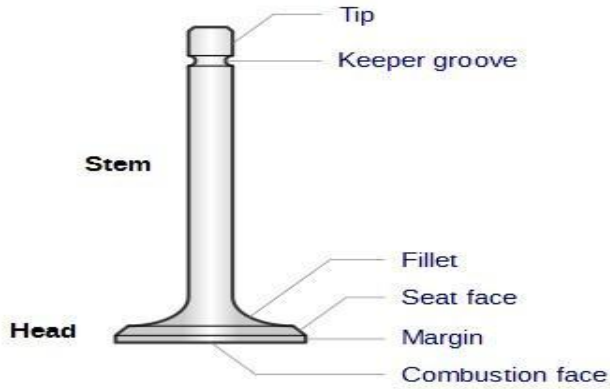
A parametric study on the effects of swirl, initial turbulence, oxygen concentration and ignition delay on fuel vaporization, mixing and combustion process. The effects of intake flow field were further investigated by varying the geometry of the intake ports the code solves the three dimension conservation equations and uses an improved RNG k-e turbulence model. The effect of swirl on emissions is not considered here. The influence of different initial condition procedures on combustion and emissions predictions in small-bore high - speed injection. The analysis was performed by using the Ansys cfx. Observed that the standard k-e model with wall function for a description of boundary layer behavior predicts the fully turbulent flow inside the cylinder.

2. INTRODUCTION TO CFD

Valve

As the airflow passes various components and stages of the intake system, different properties and characteristic of the intake charge have been modified to achieve the overall goals of the air management system.

The intake air filter ensures that air cleanliness is adequate, the charge air composition and oxygen content are controlled by introducing EGR to the intake air and the compressor and charge air cooler ensure that intake manifold pressure and temperature objectives are met and that intake charge density is within design limits. A few final aspects of air management are achieved after the intake charge exits the intake manifold and enters the cylinder. Valves or ports control the timing of air flow to the cylinder. Also, the passage between the intake manifold and cylinder can have a significant influence on the flow as it enters the cylinder and can be used to impart a suitable bulk motion and kinetic energy to the charge to support the mixing of air, fuel and intermediate combustion products in-cylinder.



About swirl

Swirl is defined as the organized rotation of the charge about the cylinder axis. It is created by bringing the intake flow into the cylinder with an initial angular momentum. Swirl is generated during the intake process in SI engines by the intake port and sub sequent by combustion chamber geometry during the compression stroke. The swirl intensity increases the tangential component of the velocity of air inside the cylinder, which aids in the mixing of fuel and air, and significantly affects the combustion and emission characteristics of SI engines.

1 Suction swirl

During suction, air is admitted into the engine cylinder in a tangential direction. The entering air is deflected by the cylinder wall. Air thereby assumes a rotary motion i.e. swirl about the cylinder axis. This swirl is called suction swirl. Helical ports produce swirl upstream of the valve and directed ports have it downstream. In SI engines, tangential entry of air is affected by one of the following methods:

1. By masking a portion of the inlet valve.
2. By angling the inlet port in the desired direction.
3. By providing a lip in the inlet port, over one side of the inlet valve.

2 Compression swirl

The combustion chamber cavity tends to modify the swirl as the piston approaches the Top Dead Centre (TDC) position during the compression process. As the piston approaches TDC the rotating air is forced into the piston bowl. The bowls have a higher swirl rate.

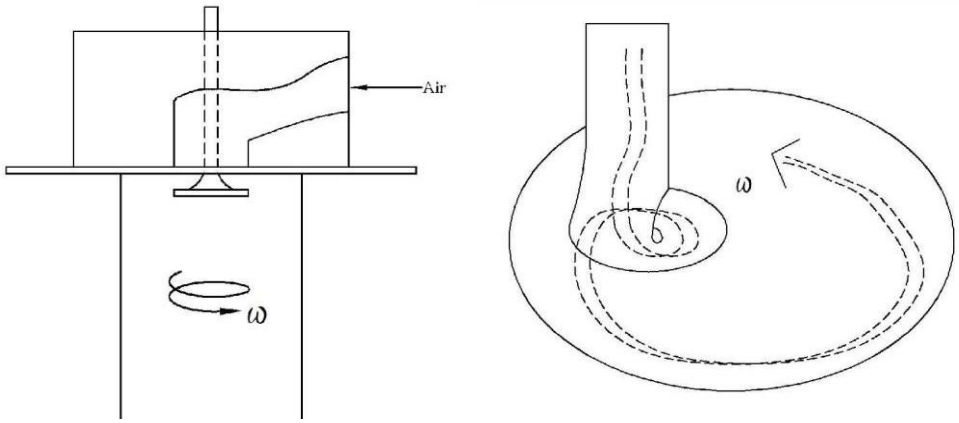


Fig 2.1. Schematic of swirl air motion

3 Swirl Valve

It is seen that the poppet valve is more effective than any other intake valve in terms of producing a high level of intake swirl. So, in order to further increase the intake swirl generation capacity of the poppet intake valve, two curve blades are incorporated on its neck.

Curve blades on the neck of the poppet valve are preferred over the conventional shrouded poppet valve since; it will provide lesser blockage to the incoming charge and hence, will result in higher volumetric efficiency the conventional shrouded poppet valve.

The computational result for high swirl induction intake valve at 72°crank angle shows that the tangential velocity produced by the incoming charge is higher compared to the tangential velocity produced by the incoming charge on using the poppet intake valve at same crank angle. From the above computational it is also 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 28.8mm from engine cylinder head which is at a higher distance from the intake valve.

The computational result at 123 crank angle for high swirl induction intake valve. The computational result for high swirl induction intake valve at 123 crank angle shows that the tangential velocity produced by the incoming charge is higher compared to the tangential velocity produced by the incoming charge on using the poppet intake valve at the same crank angle. From the above computational it is again seen that the surface at 15.3mm from engine cylinder head which is closer to the valve shows higher tangential velocity at various location compared to the surface at 61.2mm from engine cylinder head which is at a higher distance from the intake valve.

Available to make the spoiler. Further, the regulations restrict any part of the car, outstretching the front two wheels. This meant the spoiler cannot be any wider than 120 cm. A 6 cm clearance was provided to the wheel from the whole front spoiler arrangement, for it to steer safely.

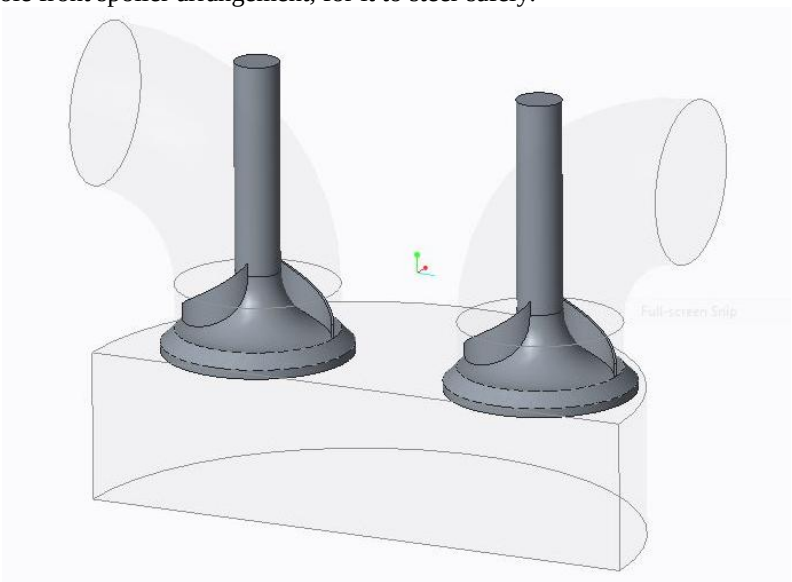


Fig 2.2

3. Methodology

The methodology applied for this research is that of CFD Analysis which is briefly explained below:

- In this analysis, we will discuss the flow simulation of the combustion chamber.
- Parameters used in the analysis are temperature and pressure.
- External flow analysis is done.
- Select the Axis.
- The fluid used is air.
- Temperature inputted is 373 K.
- The computational domain is selected to simplify the computing.
- Global goals like total pressure and frictional force are defined.
- Run the simulation.
- Get flow trajectories in terms of pressure.
- Get surface plots and cut plots.

Process detail

The methodology adopted for the present work is as follows. Flow through the intake manifold is simulated to study the in cylinder flow field during non- reacting conditions, which includes the following steps:

- Solid modeling of the intake manifold and cylinder geometry with valves.
- Mesh generation. Solutions of the governing equation with appropriate boundary conditions.
- Comparison of the simulated results with the available results in the literature.

The study is expected to explore the potential of using CFD tool for design and optimization of engine inlet valve. The commercial CFD code IC ENGINE is used for the analysis of flow. The CFD package includes user interfaces to input problem parameters and to examine the results. The code contains three elements

- Pre-Processor
- Solver
- Post Processor

Pre-processor mainly involves the creation of the basic 3D model, grid generation and fixing of the boundary conditions. Modeling and meshing are done IC ENGINE in and is exported to IC ENGINE for completing the mesh. Numerical solution techniques that form the basis of the solver perform the following:
Approximation of unknown flow variables by means of simple functions:

- Discretization by substitution of the approximations into governing flow equations and subsequent mathematical manipulations
- Solution of the algebraic equations

Partial differential equations for conservation of mass, momentum, energy, chemical species, turbulent kinetic energy and its dissipation rate are integrated over individual finite control volumes and the resulting volume integrals are transformed to their surface counterparts. The equations represent an algebraic form of the discredited conservation equations, which are solved using an iterative methodology. The pressure-velocity coupling is achieved using PISO (Pressure Implicit Splitting of Operators) algorithm. The second order differencing scheme MARS (Monotone Advection and Reconstruction Scheme) is used for the present investigation.
Post processor of the code is used for the analyses and display of results is in the following manner:

- Domain geometry and grid display
- Vector plots Line and shaded contour plots
- 2D and 3D surface plots
- Particle tracking

4. Results and Simulations

4.1. Results

RESULTS AND SIMULATIONSOMAIN	NODES	ELEMENTS
port chamber	97528	381094
port chamber0	37858	59004
port chamber1	37858	59004
port invalve1 port	5666	17076
port out the plenum	282158	404090
All Domains	461068	920268

Table 4.1.1 Mesh Information for ICE

MODEL	SETTINGS
Space	3D
Time	Steady
Viscous	Standard k-omega turbulence model

Table 4.1.2 Models

EQUATION	SOLVED
Flow	Yes
Turbulence	Yes
Energy	Yes

Table 4.1.3 Equation

TYPE	ZONES	VALUES
pressure-outlet	ice-outlet	Gauge Pressure (pascal) - 5000 Backflow Total Temperature (k) 300
Wall	wall-port-invalve1-port	Temperature (k) 300
Wall	wall-port-chamber	Temperature (k) 300
Wall	ice-cyl-port-chamber0	Temperature (k) 300
Wall	ice-cyl-port-chamber1	Temperature (k) 300
Wall	ice-cyl-port-out plenum	Temperature (k) 300
Wall	ice-cyl-port-chamber	Temperature (k) 300
Wall	ice-slip wall-out plenum	Temperature (k) 300
Wall	ice-valve-proximity-faces	Temperature (k) 300

Table 4.1.4 Boundary Conditions

VARIABLE	RELAXATION FACTOR
Density	1.000
Body Forces	1.000
Turbulent Kinetic Energy	0.750
Specific Dissipation Rate	0.750
Turbulent Viscosity	1.000
Energy	0.750

Table 4.1.5 Relaxation

PARAMETER	VALUE
Type	Coupled
Pseudo Transient	Yes
Explicit momentum under-relaxation	0.500
Explicit pressure under-relaxation	0.500

Table 4.1.6 Pressure-Velocity Coupling

VARIABLE	SCHEME
Pressure	Standard
Density	Second Order Upwind
Momentum	Second Order Upwind
Turbulent Kinetic Energy	First Order Upwind
Specific Dissipation Rate	First Order Upwind
Energy	Second Order Upwind

Table 4.1.7 Discretization Scheme

4.2. Simulations

4.2.1. Animation: velocity-magnitude on "ice_cutplane_1"

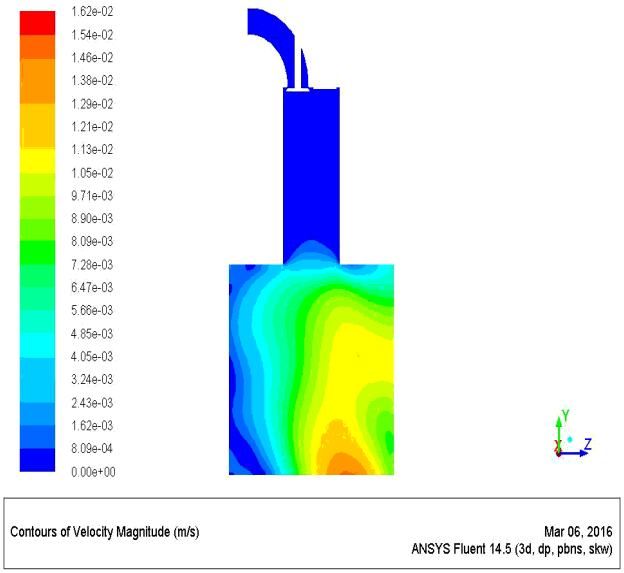


Fig.4.2.1

4.2.2 Table: velocity-magnitude on "ice_swirl_plane_1" "ice_swirl_plane_2" "ice_swirl_plane_3"

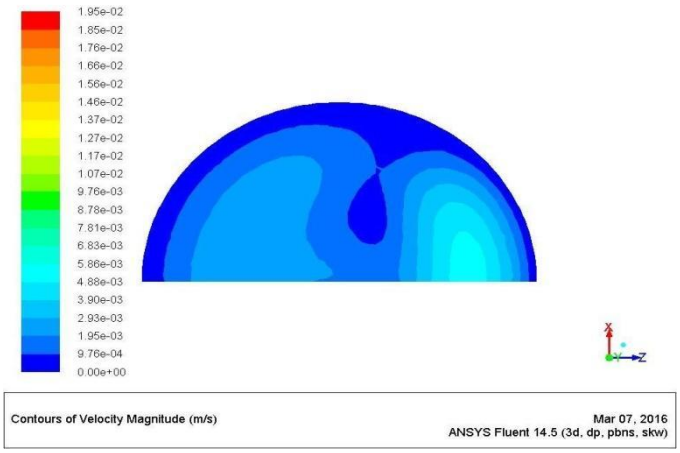


Fig. 4.2.2

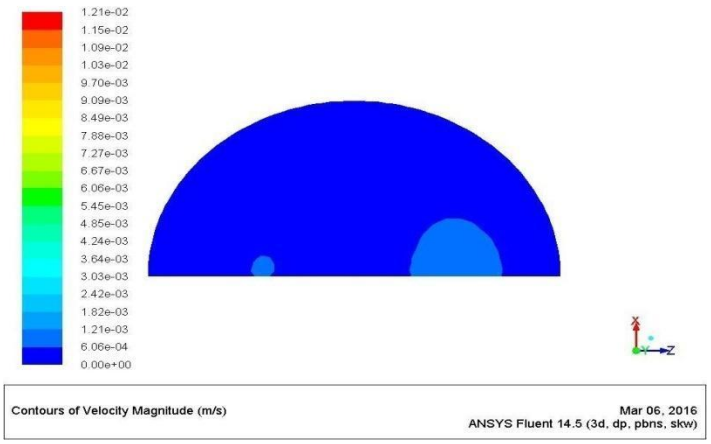


Fig. 4.2.3

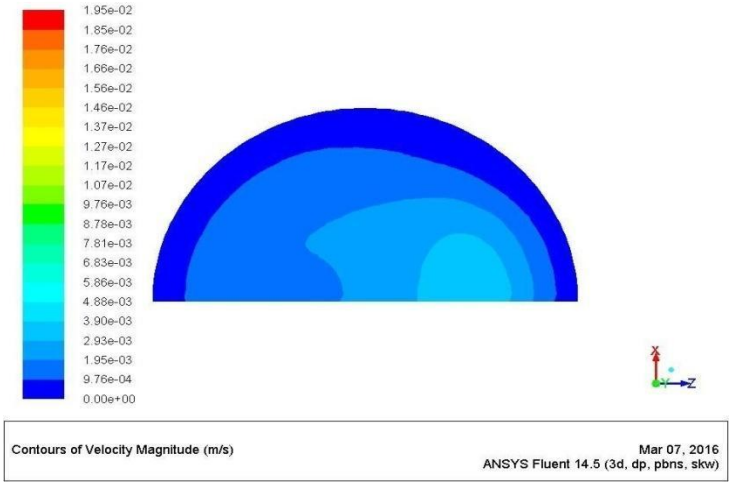


Fig. 4.2.4

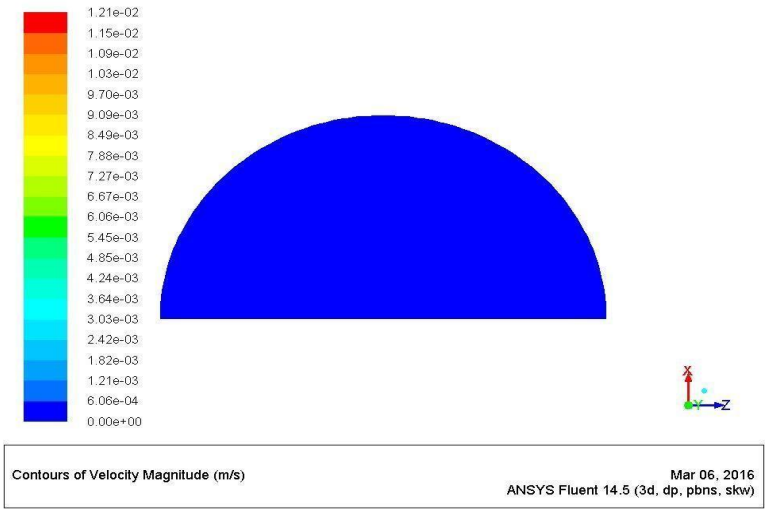


Fig. 4.2.5

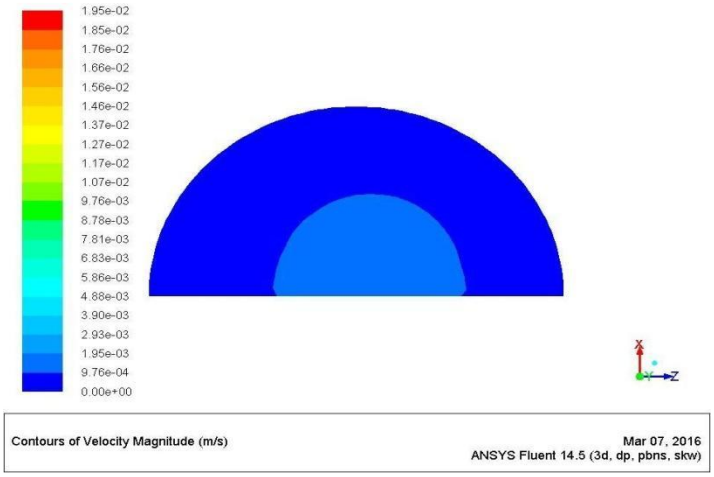


Fig. 4.2.6

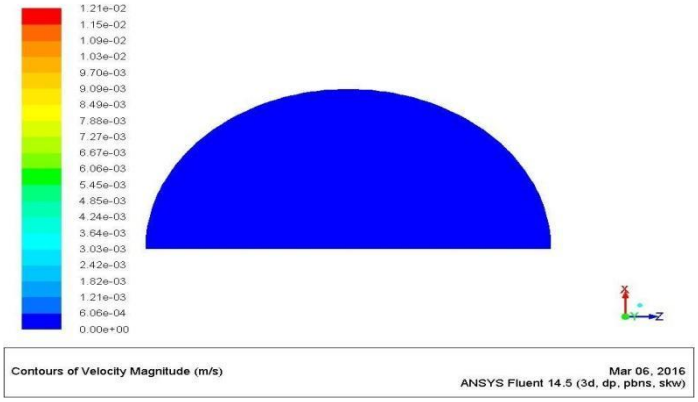


Fig. 4.2.7

4.2.3 Table: velocity-magnitude on "ice_cutplane_1"

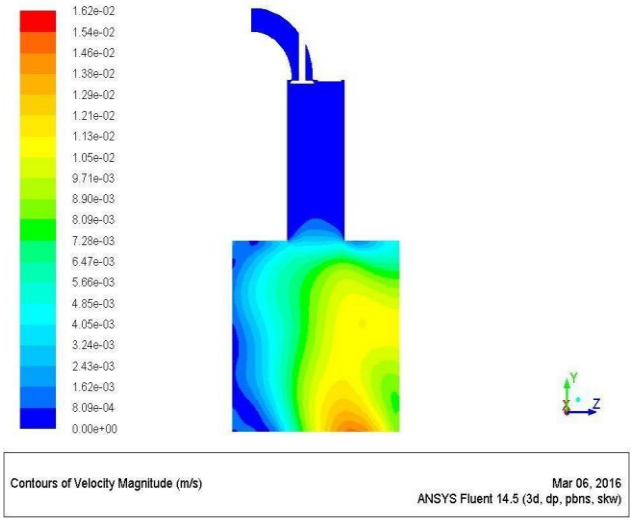


Fig. 4.2.8

4.3. Residuals

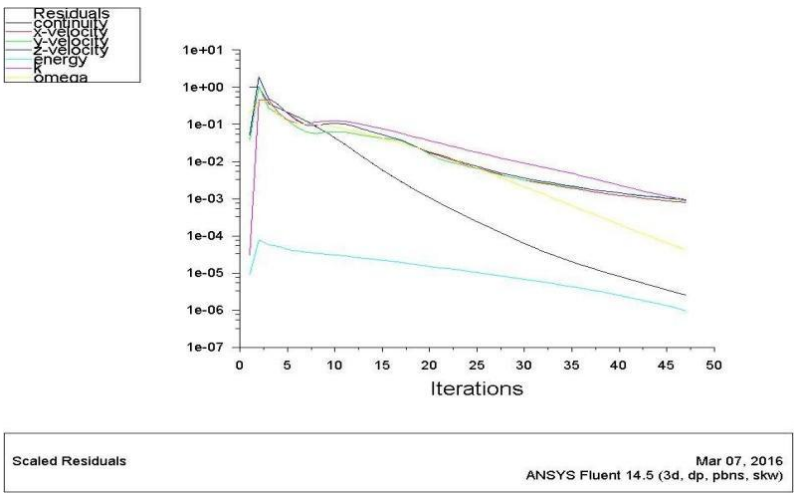


Fig. 4.3.1

4.4. Charts

Chart 1. Monitor: Mass Flow Rate (ice-outlet)

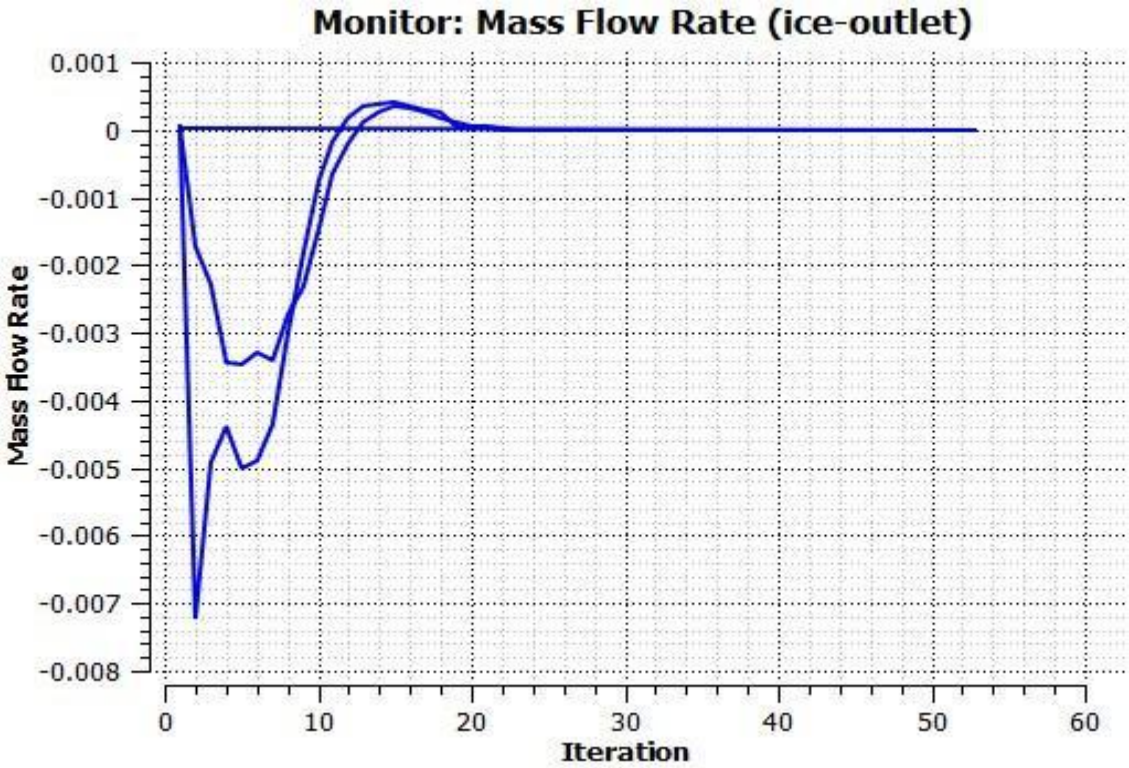


Chart 2. Monitor: Flow Rate swirl1 (ice_swirl_plane_1)

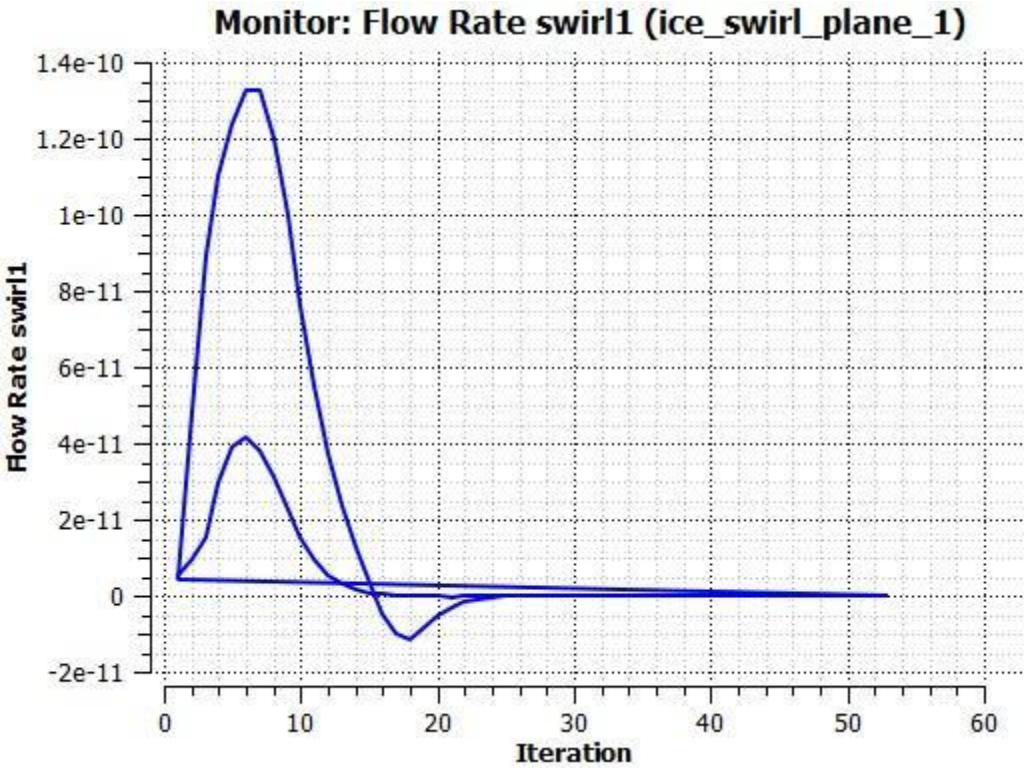


Fig. 4.4.1

Chart 3. Monitor: Flow Rate swirl2 (ice_swirl_plane_2)

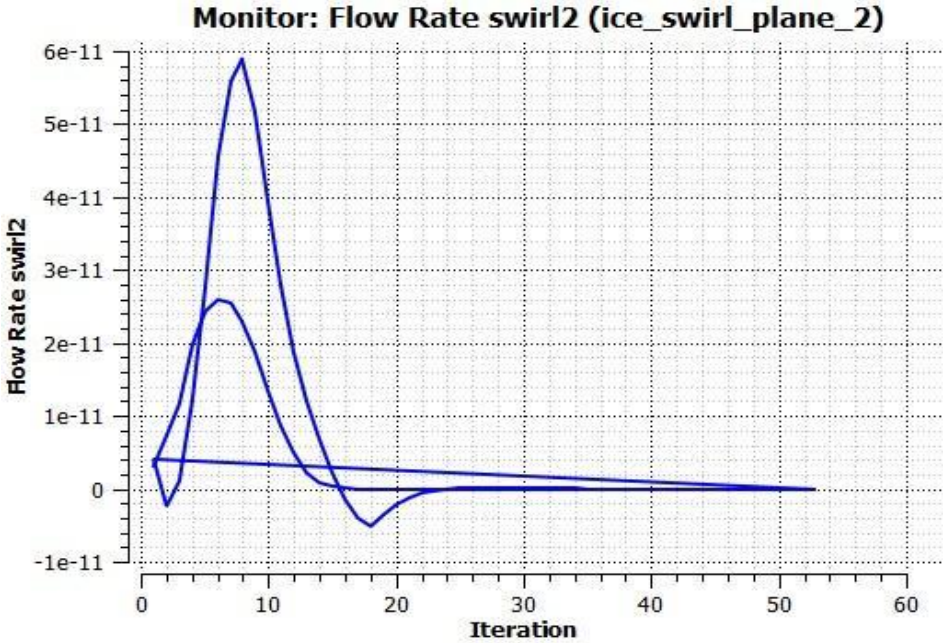


Chart 4. Monitor: Flow Rate swirl3 (ice_swirl_plane_3)

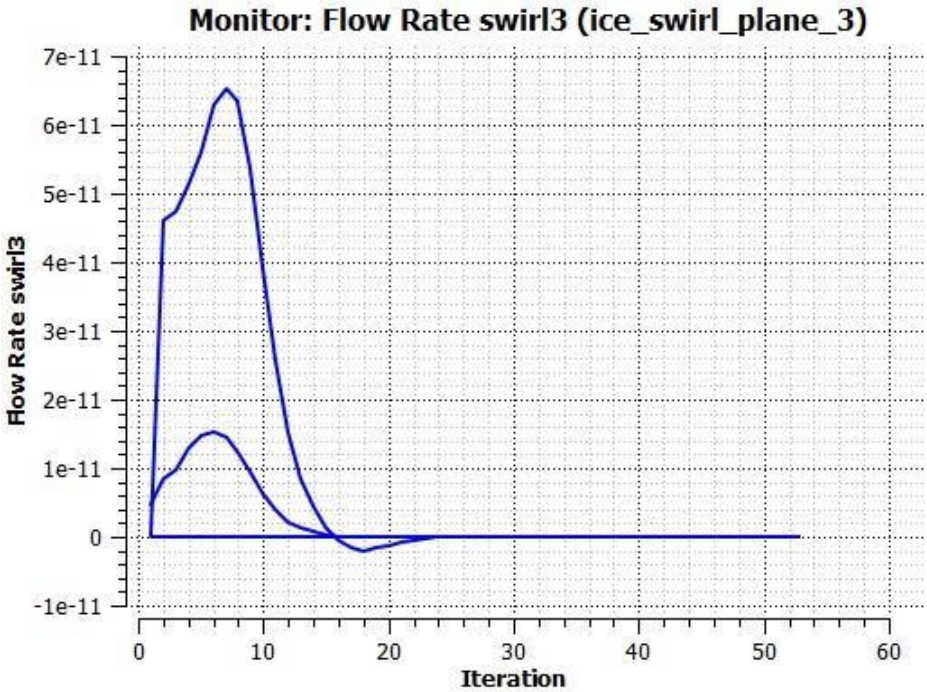


Fig. 4.4.2

Chart 5. Convergence history of Velocity Magnitude on ice-int-chamber1-outplenum (in SI units)

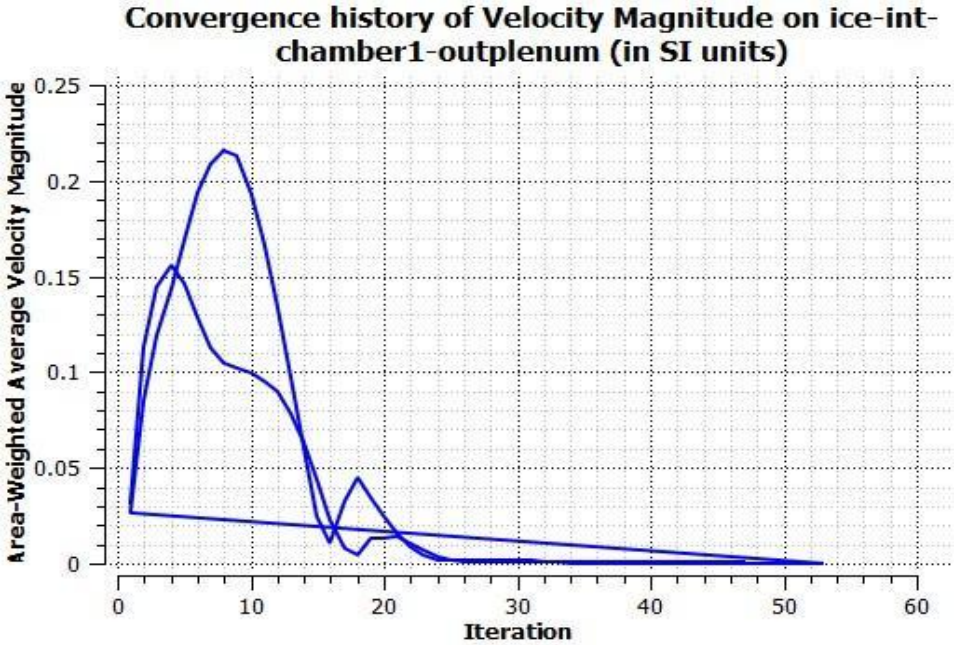


Fig. 4.4.3

5. Design Points Report

Velocity-magnitude on "ice_cutplane_1" for Design Points "DP 0"

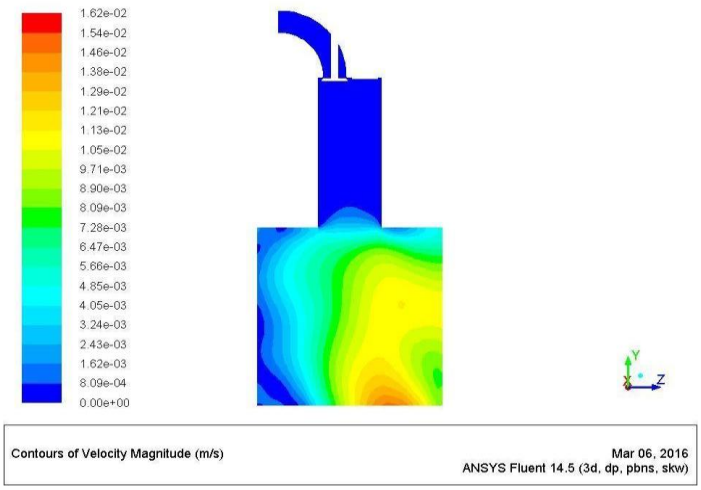


Fig 5.1

The simulation approach is carried out for the intake port to have optimized swirl torque which enhances the maximum performance of the internal combustion engine. Simulation results show the intake port gives lower swirl ratio and high flow coefficient for the better performance of the engine.

6. Conclusion

Optimization of inlet air to the engine can be done by means of inlet poppet valves. CFD simulation can advise better results. Masking of inlet valves improves swirl rate and intern brake thermal efficiency of the engine. Fins also increase the swirl rate and hence we can get better thermal efficiency. The swirl helps to improve a performance and reduce the exhaust emissions. Pollution levels are also decreased with both valves

when compared to the conventional valve. The manufacturing cost of masks and fins are less and can easily prepare.

References

[1.] V.CVS Phaneendra, V.Pandurangadu & M. Chandramouli (2012)," Performance Evaluation Of A Four Stroke compression Ignition Engine With Various Helical Threaded Intake Manifolds", International Journal of Applied Research in Mechanical Engineering.

[2.] S.L.V. Prasad and V. Pandurangadu (2013)," Reduction Of Emissions By Intensifying Air Swirl In A Single Cylinder Di Diesel Engine With Modified Inlet Manifold", International Journal of Applied Engineering and Technology.

[3.] P. Ramakrishna Reddy, K. Govinda Rajulu and T. Venkata Sheshaiah Naidu (2014)," Experimental Investigation on Diesel Engines by Swirl Induction with Different Manifolds", International Journal of Current Engineering and Technology.

[4.] Jorge MARTINS, Senhorinha TEIXEIRA & Stijn COENE (2009)," Design Of An Inlet Track Of A Small I. C. Engine For Swirl Enhancement", 20th International Congress of Mechanical Engineering, Gramado, RS, Brazil.

[5.] Benny Paul, V. Ganesan (2010)," Flow field development in a direct injection diesel engine with different manifolds", International Journal of Engineering, Science and Technology.

[6.] Suresh .Aadepu, I.S.N.V.R. Prasanth, Jarapala.Murali Naik (2014), "Design of intake manifold of IC engines with improved volumetric efficiency", International Journal & Magazine Of Engineering, Technology, Management And Research.

[7.] S. A. Sulaiman, S. H. M. Murad, I. Ibrahim And Z. A. Abdul Karim (2010),"Study Of Flow In Air-Intake System For A Single-Cylinder Go-Kart Engine" International Journal Of Automotive And Mechanical Engineering.

[8.] B. Murali Krishna and J.M. Mallikarjuna (2011), " Effect of Engine Speed on In-Cylinder Tumble Flows in a Motored Internal Combustion Engine-An Experimental Investigation Using Particle Image Velocimetry", Journal of Applied Fluid Mechanics.

[9.] D.Ramasamy, Zamri.M, S. Mahendran, S.Vijayan," Design Optimization of Air Intake System (AIS) of 1.6L Engine Adding Guide Vane", Proceedings of International

[10.] Idris Saad and S. Bari (2013), "CFD Investigation Of In-Cylinder Air Flow To Optimize Number Of Guide Vanes To Improve CI Engine Performance Using Higher Viscous Fuel", International Journal of Automotive and Mechanical Engineering.

[11.] A. Martínez-Sanz, S. Sánchez-Caballero, A. Viu, R. Pla-Ferrando (2011),"Design And Optimization Of Intake Manifold In A Volkswagen Car" ANNALS of the ORADEA UNIVERSITY. Fascicle of Management and Technological Engineering.

[12.] F. Payri, J. Benajes, X. Margot, A. Gil (2004)," CFD modeling of the in-cylinder flow in direct injection Diesel engines", Elsevier Computers & Fluids 33.

[13.] S. Siva, Dr.M.Subramanian, And K.Sivanesan (2013), " Numerical Simulation Of 3d Kirlosker Tv-1 Model Engine Cylinder For Cold Flow", International Journal of Engineering Science and Technology.

[14.] Jay V. Shah and Prof. P.D.Patel (2014), " Experimental Analysis of Single Cylinder 4-Stroke Diesel Engine For the Performance and Emission Characteristics at Different Inclinations Of The Intake Manifold", International Journal of Scientific Research & Development.

[15.] Y.K. Loong and Salim M. Salim (2013)," Experimentation and Simulation on the Design of Intake Manifold Port on Engine Performance", EURECA 2013.

