STUDIES ON THE EFFICACY OF HELICAL PORT DESIGN USING COMPUTATIONAL FLUID DYNAMICS

by

Isaac Premkumar JOSWA LAZARAS [a*], ***Sudhagar SUBRAMANIAN*** [b], ***Lakshmanan PANDIYAN*** [c], ***Hariharan KONDIN BALAKRISHNAN*** [c], ***and Ranjith Kumar MURUGESAN*** [c]

[a] Department of Mechanical Engineering, JCT College of Engineering and Technology, Pichanur, Coimbatore, Tamilnadu, India
[b] Department of Mechanical Engineering, University College of Engineering, Dindigul, Tamilnadu, India
[c] Department of Mechanical Engineering, K. Ramakrishnan College of Technology, Trichy, Tamilnadu, India

Original scientific paper
https://doi.org/10.2298/TSCI190709441J

In-cylinder gas motion is the essential parameter that enhances fuel-air mixing and increases the combustion rate and governs the performance and emission of the engine. It also has significant impact on heat transfer. Angling the intake port into various angles is the one of the method to enhance swirl motion inside the combustion chamber. In this project, direct port is converted into helical port to analyze the air motion inside the engine cylinder. In order to analyze the effect of helical port design the various flow parameters such as swirl ratio, flow coefficient, and turbulent kinetic energy are calculated. By angling the intake port, swirl generation during suction stroke is comparatively larger than direct port. A commercial CFD software STAR-CCM is used to analyze the in-cylinder air motion.

Key words: *computational analysis, naturally aspirated engine, top dead centre*

Introduction

The most important function of Diesel engine combustion chamber is to provide proper air and fuel mixture in short span of time. In order to get this, an proper air movement named as swirl is provided to generate high relative velocity between the air and fuel droplet [1]. During the air jet is injected into the combustion chamber, the spray cone getting disturbed due to air motion inside that can be guided by the intake port, makes it necessary to study the helical port design in detail. The important role of in-cylinder air motion starts from the earlier start of the engine cycle. During the suction stroke, the incoming air provides flow structures within the cylinder, which leads to determine the extent of mixing between the fresh charge and the residuals, and internal and external heat transfer rates [2]. The flow field plays a major role in preparation of charge and conditioning for ignition. During the combustion processes, the flow continues to exert its effect on heat transfer and thermal stress levels. The effects of various geometrical parameters of helical port on swirl performance was evaluated with CFD. The computational analysis approach was used for analysis the flow in the intake port [3]. The port performance was analyzed experimentally with the help of steady flow rig. Modeling is done using

* Corresponding author, e-mail: isaacpremkumar007@gmail.com

PRO-E software. Engine test was carried out in a naturally aspirated three cylinder, four-stroke having bore of 100 mm, a stroke of 100 mm. The parametric study concluded that, the swirl ratio increases with a decreasing T/R value. There is a correlation between the swirl ratio, R, and the throat height, T. In the case of lower T/R, it clearly shows that most of the streamlines of the incoming air follow strikes the cylinder wall and generate strong swirl. In other way, Streamline is directed towards the axis of the cylinder, when the T/R value is higher. That leads to the lowering the swirl ratio inside the engine combustion chamber. The CFD was used as a tool to optimize the suction port design for better volumetric efficiency of the engine and validated the computational result with flow bench test rig [4]. The calculation was carried out with different valve lift of the engine [5]. The flow coefficient and swirl ratio were measured from it. Flow bench test rig is taken for the experimental investigation. Flow through the intake port is considered as a steady flow for analysis. Constant pressure difference maintained the inlet mouth and in cylinder chamber [6]. That pressure difference act as gradient for make the flow from atmosphere to engine combustion chamber. Flow bench controller is used for control the system pressure. Swirl ratio was calculated by paddle wheel rotation. For more accuracy test is conducted by experimental as well as computational flow analysis [7, 8]. Their compared results showed that swirl ratio deviated at higher valve lift, and volumetric efficiency increased 10% by increasing the flow coefficient at high valve lift. Swirl ratio plays a key role in the engine intake flow.

Methodology

Computational flow analysis of 3-D in cylinder cold flow analysis was conducted in this work, to study the behavior of intake air-flow by STAR CCM+ software. It includes developing the direct and helical intake ports using STAR CCM+, Before starting the analysis, initial and boundary conditions are assigned for the model. Then, computational flow analysis is performed by software. The model was generated with a consideration of normally aspirated Diesel engine. After the simulation different flow parameters such as turbulent kinetic energy, pressure, velocity and swirl ration is calculated manually and simulation. In order to assess the intake port performance, flow coefficient and swirl ratio as 2-D parameters to be considered and measured the flow capacity and the intensity of swirl formation also plotted in this work.

Engine geometry and computational details

The intake port geometry studied in this numerical investigations for compression ignition engine, includes intake port and intake valves and combustion chamber. On the top of the engine cylinder, intake and exhaust port are situated, each in quarter section of the circular cylinder [9]. For the simulations, exhaust valve is considered closed condition, only the intake port is kept open for in cylinder cold flow analysis. In this work, computational analysis of compression ignition engine is taken for analysis. Maximum rate power of the engine is about 4.41 kW. Modeling is done with the help of STAR CCM+ software, which is commonly used for engine intake flow analysis. The relevant technical specification of an engine is given in tab. 1.

Preprocessing – computational Model

The geometry used in this work is described in figs. 1 and 2. In the first case, intake port is designed with bottom slop angle, α, and helix slope angle, β, as zero. In helical port α is kept as 45° and β is taken in three different angle for increase the intake flow charge. The β angles are 120°, 150° , and 180°. In the pre-processing steps, computational model is generated and discretization is done using STAR CCM+ software. Discretization details for direct port are given in tab. 2.

Table 1. Technical specification of an engine

Parameters	Values	Parameters	Values
Make and model	Kirloskar & TAF1	Rated speed, [rpm]	1500
No of cylinders	1	Lub. oil consumption	0.8% of specific fuel consumption
Bore × stroke, [mm]	87.5 × 110	Type of fuel injection	Direct injection
Cubic capacity, [cc]	662	Maximum intake valve lift, [mm]	11
Compression ratio	17.5:1	Intake valve head diameter, [mm]	34.10
Rated output [kW], (hp)	3.7 (5)	Intake valve stem diameter, [mm]	7.9

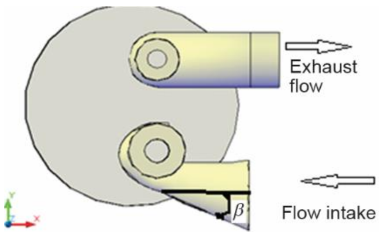

Figure 1. Top view of helical port geometry

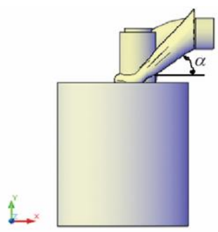

Figure 2. Front view of helical port geometry

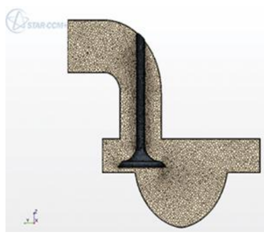

Figure 3. Computational mesh of direct intake port

Preprocessing – discretization

The surface and volume mesh were created in STAR CCM+. The volume mesh consists of polyhedral element. Figure 3 shows the computational mesh of the direct intake port. Discretization details for helical port of three different angles are given in tab. 3. Number of cells and number of faces generated in the geometric model is a polyhedral element type with a base size of 1 mm.

In this simulation, *k-ε* model used as a turbulence model. Mean piston speed calculated as 5.5 m/s and it is given as grid velocity for morphing of piston from TDC to BDC. The inlet velocity of air at intake manifold is taken as stagnation. Pressure at the inlet is taken as 1 bar (atm). Other boundaries are taken as wall. Maximum valve lift given as 11 mm.

Table 2. Discretization details of direct port

Parameters	Values
Number of cells	294350
Number of faces	490136
Element type	Polyhedral
Base size	1 mm

Table 3. Discretization details of helical port

Case	No. of cells	No. of faces
I ($\beta = 120^o$)	301936	905808
II ($\beta = 150^o$)	312655	937965
III ($\beta = 180^o$)	323766	971298

Pre processing – boundary conditions

In this simulation, *k-ε* model used as a turbulence model. Mean piston speed calculated as 5.5 m/s and it is given as grid velocity for morphing of piston from TDC to BDC. The inlet velocity of air at intake manifold is taken as stagnation. Pressure at the inlet is taken as 1 bar (atm). Other boundaries are taken as wall. Maximum valve lift given as 11 mm.

Solving methods

The governing equations namely *x, y*, and *z* components of velocities, *k-ε* equations are solved. The *k-ε* model is taken as a turbulence model.

Results and discussion

In this work, direct intake port is converted into helical port with three different helix slope angle and analyzed with the help of STAR CCM+. Port performance is evaluated by flow parameters like swirl ratio, velocity, and pressure and turbulent kinetic energy. Both direct and helical port parameters are compared and graphs plotted. Swirl ratio and velocity of the air increases as the helix slope angle increase. Due to increase in angular momentum of inlet charge during the suction stroke. The pressure drop across the port-valve assembly varied with helix slope angle. It is also maximum in the case of higher helical slope angle port.

Velocity profile of helical ports

Figure 4 shows the three different velocity profile of helical ports. The port which having helix slope angle 180° has achieved maximum velocity at the intake valve due to the angular momentum generated in the helical intake port.

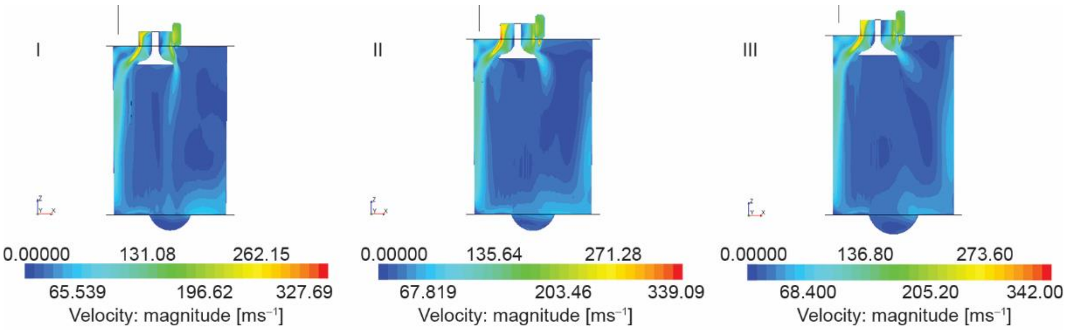

Figure 4. Velocity profiles of helical ports; Case I: β = 120°, Case II: β = 150°, Case III: β = 180°

In the Case III, maximum velocity of intake gas is achieved as 342 m/s in the throat section of the intake port. Where the cross-section area of the port is minimized due to intake valve present in the throat that causes to increase the maximum velocity at the throat.

Velocity swirl disk

Figure 5 shows the velocity vector swirl disk of helical port (β = 180°) at three different length of stroke and it shows the how the air get directed from the helical port to the combustion chamber. Velocity decreases as the length of stroke increases. Maximum velocity nearer the valve due to angular momentum of inlet charge. Swirl ratio of the intake flow is calculated based on the following formulae, swirl disk is placed with a stroke length of 11 mm, 55 mm, and 110 mm of the displacement length. Maximum swirl is achieved in the Case I. Because swirl generated in this work is fully induced swirl. It keeps on decreasing with increasing the stroke length. Air particle is trapped in the combustion chamber with radially inward direction. At the end of suction stroke, swirl present in the chamber in the range of 2 to 70.

$$\text{Swirl ratio} = \frac{\text{Rotational speed of swirl}}{\text{Engine speed}}$$

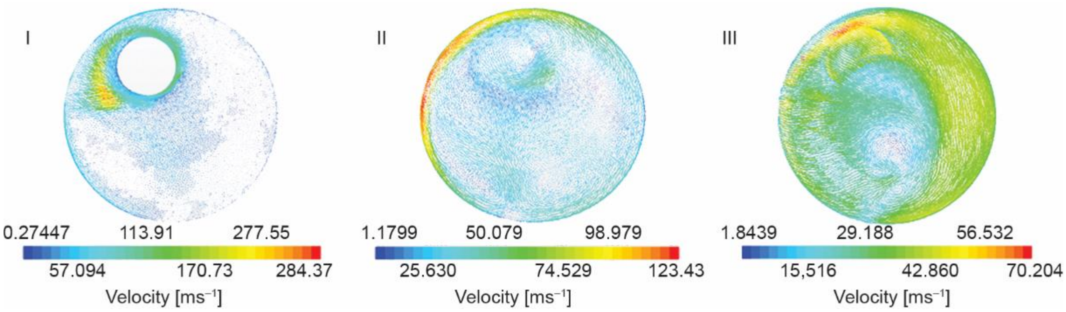

Figure 5. Velocity swirl disk at different length of stroke; SL – 11 mm, SL – 55 mm, SL – 110 mm

Swirl ratio of helical ports

The swirl ratio increases as the helix slope angle increases. It is because the angular momentum generated in the helical port. Swirl ratio is achieved in this case is 3.2 at the centre of the piston bowl which, is shown in fig. 6. Dimensionless parameter used to quantify rotational motion with in the cylinder. Swirl generated in the intake usually persists through the compression, combustion and expansion. During intake process it increases then decreases after reaching the TDC due to viscous drag.

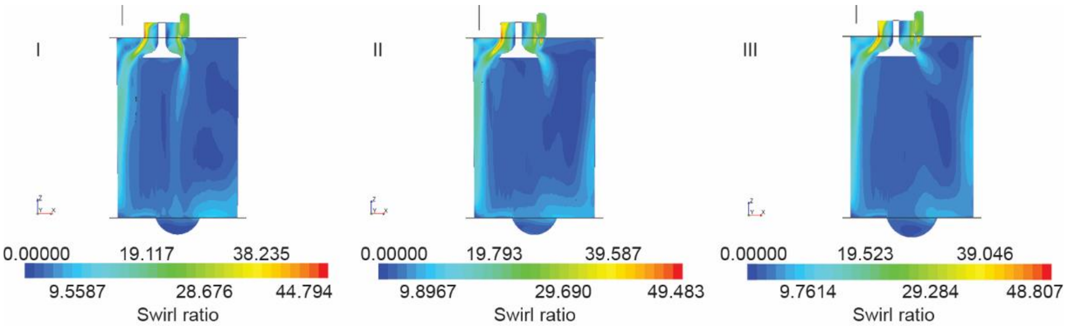

Figure 6. Swirl ratio of helical ports; Case I: β = 120°, Case II: β = 150°, Case III: β = 180°

Swirl ratio comparison

From the CFD analysis, swirl ratio during the suction stroke is calculated with the assumption of velocity V = 16 m/s and angular velocity of swirl is taken as 365.17 rad per second. Engine speed is taken as 1500 rpm. Figure 7 shows the swirl ratio comparison of direct and helical ports. From the graph, Maximum swirl ratio is achieved with the helix angle of 180°due to increasing the angular momentum of the intake charge.

Streamline path of helical ports

The path of a particle in a fluid relative to solid body is traced by a streamline curve generated by STAR CCM+ software. Velocity with a magnitude of 384 m/s is generated in the intake throat. When the piston at bottom most of the cylinder, the velocity of intake charge is high in the axis of the combustion chamber, Which leads to maximize the atomization, and evaporation of fuel particle in the further stroke. It helps to achieve better combustion and simultaneous reduction of unburned gases in the engine cylinder, fig. 8.

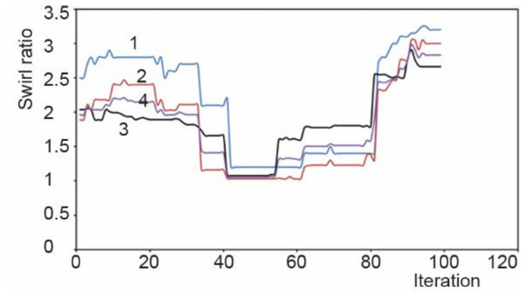

Figure 7. Swirl ratio comparisons of direct and helical ports;
1 – swirl ratio (for angle 180°),
2 – swirl ratio (for angle 150°),
3 – baseline,
4 – swirl ratio (for angle 120°)

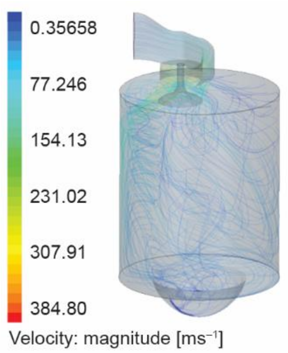

Figure 8. Streamline path of intake charge during suction stroke

Conclusion

In is concluded that, direct port of an internal combustion engine is changed into helical port in order to provide better swirl motion in the combustion chamber. Velocity of air inside the combustion chamber is increased due to helical inlet port which induce the angular momentum of inlet charge and swirl ratio is improved in helix angle 180°. So better port performance is achieved using helical port.

References

[1] Sabale, S. K., Sanap, S. B., Design and Analysis of Intake Port of Diesel Engine for Target Value of Swirl, *American Journal of Mechanical Engineering, 1* (2013), 5, pp. 138-142
[2] Winterbone, D. E., Pearson, R. J., *Design Techniques for Engine Manifolds, Wave Action Methods for IC Engines,* Society of Automotive Engineers Inc. Warrendale, Penn., USA, 2001
[3] Saravankumar, P. T., *et al.*, Ecological Effect of Corn Oil Biofuel with SiO_2 nanoadditives, *Energy Sources, Part A: Recovery, Utilization, and Environmental Effects, 41* (2019), 23, pp. 2845-2852
[4] Vivekanandan, M., *et al.*, Pressure Vessel Design using PV-ELITE Software with Manual Calculations and Validation by FEM, *Journal of Engineering Technology, 8* (2019),1, pp. 425-433
[5] Pradeep Mohan Kumar, K., *et al.*, Computational Analysis and Optimization of Spiral Plate Heat Exchanger, *Journal of Applied Fluid Mechanics*, *11* (2018), Special issue, pp. 121-128
[6] Mohamed Niyaz, H., Dhekane, A. S., Twin Helical Intake Port Design optimization and Validation by using CFD Analysis, *International Journals of Emerging Technology and Advanced Engineering, 4* (2014), 4, pp. 454-462
[7] Avudaiappan, T., *et al.*, Potential Flow Simulation through Lagrangian Interpolation Meshless Method Coding, *Journal of Applied Fluid Mechanics*, *11* (2018), Special issue, pp. 129-134
[8] Mohamed Niyaz, H., Dhekane, A. S., Design Optimization of Intake Port in Diesel Engine by using CFD Analysis, *International Journals of Engineering Research and Technology, 2* (2013), 11, pp. 2902-2912
[9] Srinivasan, R., *et al.,* Computational Fluid Dynamic Analysis of Missile with Grid Fins, *Journal of Applied Fluid Mechanics*, *10* (2017), Special issue, pp. 33-39

Paper submitted: July 9, 2019
Paper revised: August 11, 2019
Paper accepted: October 1, 2019

© 2020 Society of Thermal Engineers of Serbia.
Published by the Vinča Institute of Nuclear Sciences, Belgrade, Serbia.
This is an open access article distributed under the CC BY-NC-ND 4.0 terms and conditions.