Journal of Multidisciplinary Engineering Science Studies (JMESS)
ISSN: 2912-1309
Vol. 2 Issue 2, February - 2016

Application Of Computational Fluid Dynamics Code On Flow Process Through Inlet Port Of Internal Combustion Engines

Ajoko, Tolumoye John
Department of Mechanical/Marine Engineering,
Faculty of Engineering
Niger Delta University
Wilberforce Island, Bayelsa State, Nigeria
johntolumoye@yahoo.co.uk

Ogbonnaya, Ezenwa Alfred
Department of Mechanical/Marine Engineering,
Faculty of Engineering
Niger Delta University
Wilberforce Island, Bayelsa State, Nigeria
ezenwaogbonnaya@yahoo.com

***Abstract* — This paper aims at emerging steps to design more efficient Internal Combustion Engine (ICE) compared with what exists presently. To accomplish this aims, simulation with Computational Fluid Dynamics (CFD) code integrated with ANSYS to obtain very useful results using a modelled engine. The effectiveness of the tools cannot be over emphasized because they understood the fluid dynamics of most engines and is capable of evaluating the engine geometry and its mechanism both graphically and numerically. Also, the simulated results for the model under study was examine with comparative analytical test showing graphically display of swirl ratio and flow coefficient against lift values. Dynamic pressure difference of 0.031783Pa across the model after 63-iterations was evaluated. This attests fluid flow from one region of the inlet port to another. Also vector plot of velocity for high lift was simulated and results confirms the reliability of the tools because the CFD calculations verifies that swirl ratio and flow coefficient are good enough for the testing of fluid dynamics fundamental equations. Also results attests that the used application is capable to identify the region of the IC engine port with more flow. Therefore, test conducted with the codes were visible and reliable hence study was realistic.**

Keywords— CFD, Dynamic Pressure, Flow Coefficient, Flow Process, ICE, Inlet Port, Solidworks, Swirl Ratio, Valve Lift.

I. Introduction

Over the decades ICEs emit toxic and hazardous substances from unburned hydrocarbon fuels capable of depleting the ozone layer. This caused negative emission pose great danger to human lives and the environment. In view of these challenges, scholars have contributed enormously to the improvements of engine efficiency and power performance by the increase of in-cylinder flow process using optimized cylinder head, port and valve timing and subsequent reduction of flow coefficients for homogeneous mixture formation [1,2]. The desire to overhaul the ICEs to improve the fuel economy, heat trapping ability in the exhaust system to reduce the emissive substances and maintaining an increased engine performance such as rise in specific power, torque of engine, and increase in load during test cycle [3] is one task of researchers. Thus, the overall engine performance, development and optimization to control power and exhaust emission, consumption of fuel, etc. lies in the design of in-take port of ICEs [4].

Hence, a reviewed literature reveals that the flows in ICEs in-take port can be characterized by swirl, tumble and compression in the cylinder [5]. Also, study reveals that swirl and tumble flows in the engine cylinder are rotations in the horizontal (parallel flow) and vertical (perpendicular flow) plane respectively which improves the performance of the engine. Hence it is vital for the development of an ICE with high compression ratio to obtain high turbulence intensity and lean burn combustion [6,7]. This confirmation is also stated in a well presented paper that swirl and tumbling flow structure is more stable than other large scale in-cylinder flows. They may break up later in the cycle giving higher turbulence during combustion [8]. Though, according to Murali and Mallikarjuna, tumble flows are more crucial for proper mixing condition of air-fuel ratio for high flame propagation rate in direct injection spark ignition engines.

In a similar contribution by a scholar, swirl is described as a measurable parameter responsible for heat generation – emissions, engine efficiency – performance and influences the rate of fuel mixture in ICEs. This is made possible by the interaction of swirling field in contact with a spray via flow field variations [9]. Therefore, it is proper to have a well-designed intake manifold and cylinder head to reduce flow resistance and increase the swirl. This is so as the swirl procure adequate fuel-air mixing rate. This is done by shaping and contouring the intake manifold, valve ports, and by the use of shrouded intake valve. Meanwhile, air swirl is generated with the support of a suitable inlet port and it is amplified at the end of the compression stroke by forcing the air towards the cylinder axis into the bowl-in-piston combustion chamber [10].

A better understanding of the engine in-cylinder fluid dynamics, fuel spray behaviour, induction – generated swirl will definitely be helpful in meeting the challenges of the ICEs. Although, studies have shown that the nature of the swirling flow in an actual engine is extremely difficult to determine [11]. To this response, different studies have also been carried out to proffer solutions in the determination of flow process via ICEs inlet ports. According to open literature, simulation tools are convenient and less expensive compared to the experimental process for the designing and optimizing the flow field in the engine because the CFD understands the fluid dynamics of ICEs in respect to their geometric design of parts such as port, valves, pistons and engine parameters like valve timing and fuel injection [5,12,13,14]. Thus, the general thermodynamics study of ICEs is to improve their performance characteristics and subject it friendly to the environment. Evaluated study confirms Large-Eddy Simulations tool capable of estimating the steady flow in the port-valve-cylinder system of engines [15, 16].

Therefore, this paper presents CFD simulation of flow process in the inlet port of ICEs with the aid of Ansys Fluent tool which generates accurate results of swirl flow and flow coefficient at the different lift conditions. Also the applied application code was used to generate vector plot of velocity which validates the fluid flow process in the port under study. The significance of the study cannot be over emphasized because the layout of the intake ports of the engine where fluid flow process is analyzed is one major concern. However, study highlights and analyses a new concept for intake port design to meet the existing and future challenges of internal combustion engines.

II. MODELLING OF IC INLET PORT

Computer Aided Design (CAD) has ensure that design models can be produce at ease while capturing the actual design configuration of a product or machine component. There are several CAD tools such as Auto-Cad, Rhinoceros, Solidworks, Iron CAD etc. For this work, SolidWorks mechanical automated software is used for the modeling of the intake port. Solidworks is chosen due to its encompassing features. Solidworks is a feature based parametric design software and unlike other CAD tools, it embodies an environment for computer aided studies (simulations and analysis). Hence, Solidworks model of the intake port is first simulated in Solidworks Flow simulation environment. Table 1 summarizes the properties of the ICE whose intake port is modelled as shown in figures 1 and 2 below, and studied in this work.

Table 1. Engine Specifications and Calculation parameters

Parameter	Specification
Bore by Stroke	95mm × 99mm
Compression ratio	09:01
Max power at WOT	12.2BHP at 4950RPM
Intake Valve diameter	42mm
Maximum Intake valve lift	12mm
Piston Cavity	Flat
Exhaust Valve Opening	64^o BBDC
Exhaust Valve closure	5^o ATDC
Intake Valve Opening	5^o BTDC
Intake Valve Closure	60^o ABDC
Fuel	C_8H_{18}

Fig.1. Solidworks Computational Model

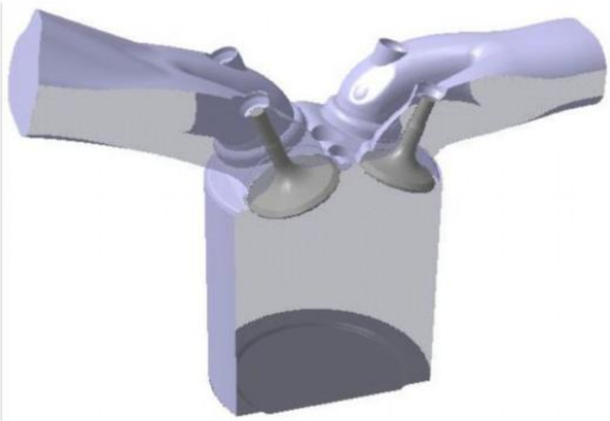

Fig.2. Section View of the Computational Model

III. MODELLING PARAMETERS

In order to carry out an accurate modeling process with the modeling tool, some performance parameters for the intake port are analyzed with the aid of the governing equations below for the inlet port as expressed in equations (1) to (10).

$$\alpha_k = \frac{m_{std}}{m_{theor}} \quad (1)$$

$$m_{std} = v \times \frac{p_{std}}{RT_{std}} \quad (2)$$

$$m_{theor} = A\rho_s C_s \quad (3)$$

$$A = \frac{\pi}{4} D_{cyl}^2 \quad (4)$$

$$\rho_s = \frac{P_1}{RT_{std}} \left[\frac{P_2}{P_1}\right]^{1/k} \quad (5)$$

$$C_s = \sqrt{\frac{2K}{k-1} RT_{std} \left[\left[1 - \frac{P_2}{P_1}\right]^{\frac{k-1}{k}}\right]} \quad (6)$$

$$Swirl\ Ratio = \frac{Circumferential\ velocity\ (C_u)}{Axial\ Velocity (C_A)} \quad (7)$$

$$C_A = \frac{V_{real}}{D_{cy}^2 \times \frac{\pi}{4}} \quad (8)$$

$$V_{real} = V \times \sqrt{\frac{\rho_{std}}{\rho_{real}}} = V \times \sqrt{\frac{P_{std} \times T_{amb}}{T_{std} \times P_{amb}}} \quad (9)$$

$$P_{stagnation} = \frac{1}{2}\rho V^2 + P_{static} \quad (10)$$

IV. CFD Boundary Conditions Simulation of ICE Inlet Port

The boundary Inlet and Outlet conditions were set as multiphase flow since fuel and air fluids are observed in the port. The fuel is defined with a mass flow rate of 0.0325kg/s, and the valve is oriented such that we have low, medium and high lift at the inlet. Thus at outlet; it is assumed that the static pressure is equivalent to atmospheric pressure condition and is given as boundary condition and the walls are assumed as adiabatic and no-slip. Tables 2 and 3 show flow coefficient and swirl ratio obtained from mathematical analysis with the use of the above governing equations and CFD simulation respectively. However, some assumptions were considered for the thermodynamics properties of the fluid (C_3H_{18}). They are such as fluid density (703Kgm^{-3}), vapour pressure (1.47KPa), test pressure and temperature for diesel engine (5 – 7Pa) and (200 – 300)℃ respectively, gas constant R of 8.314Kg/kmol [17,18]. In order to obtain accurate parameters Mean paddle wheel diameter (D_{MFL}) is adjusted between (1 – 5%) of the original value to have corresponding evaluating value of the rotational speed of the paddle wheel which is depending variable to the swirl ratio.

Table 2. Discharge Coefficient and Swirl Ratio With Mathematical Approach

Lift	Swirl Ratio	Flow Coeff. α_k
Low	0.867	0.2591
Medium	0.787	0.4411
High	0.806	0.5042

Table 3. Discharge Coefficient and Swirl Ratio With CFD Simulation

Lift	Swirl Ratio	Flow Coeff. α_k
Low	1.042	0.2361
Medium	0.914	0.4657
High	0.724	0.6015

V. Result Presentation

CFD simulation was carried out on the intake flow and results presented are both analysis from solidworks and Ansys Fluent CFD simulation. Results obtained are of two and three dimensional models. The pressure difference across the computational domain for the two dimensional simulations with solidworks flow is calculated in terms of the dynamic pressure difference and is presented in figure 3. The dynamic pressure is a critical illustration of pressure difference as fluid flow from one region to another; thus for the two dimensional calculation, Prandtl mixing length hypothesis is actually applied to the momentum and energy equation so as to solve for the energy transport across the model and is presented in figure 4.

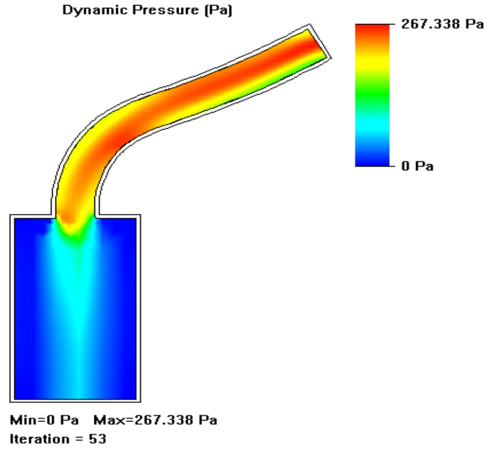

Fig.3. Dynamic Pressure across the model

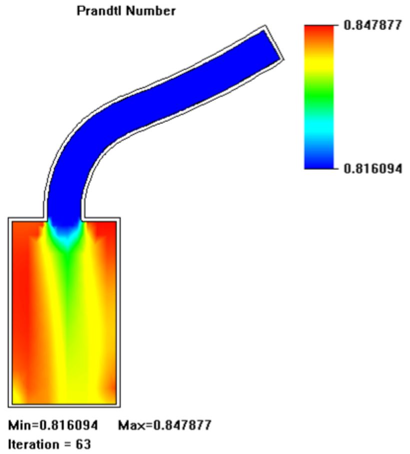

Fig.4. Prandtl Number plot across the 2D model

In order to estimate the swirl ratio from different flow parameters across the model a user defined function is applied and table 3 summarizes the result for different lift conditions. However, a CFD simulation and mathematical analysis result comparison of swirl ratio and discharge coefficient was carried out and graphical illustrations presented for the different lift conditions in figures 5 and 6 respectively. The vector plot of velocity for high lift is presented in figure 7. In demonstrates fluid flow from inner and outer diameters of the port which flows in opposite directions. This display of flow explains the position of the swirl inside the engine cylinder as discussed in the subsequent section of this paper.

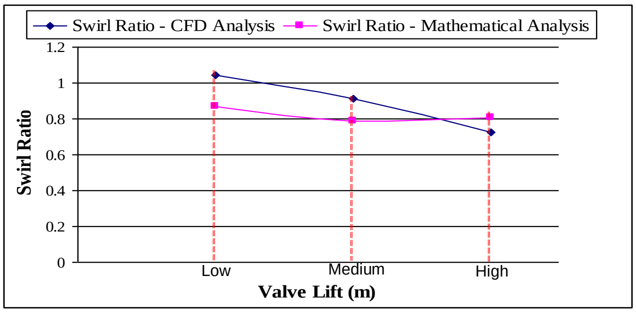

Fig.5. Comparison of Swirl ratio at different valve lift for CFD and Mathematical Analysis

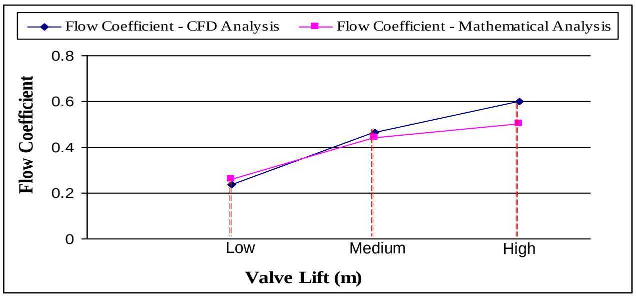

Fig.6. Comparison of Flow Coefficient at different Lift for CFD and Mathematical Analysis

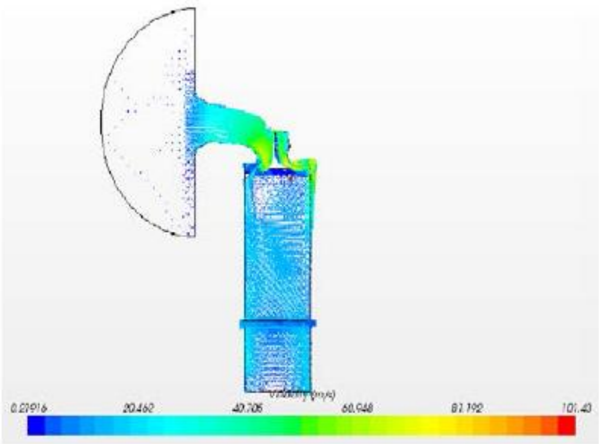

Fig.7. Vector plot of velocity for High Lift

VI. Discussion of Result

The demonstration of result in figure 3 shows that minimum pressure difference across the computational domain for the two dimensional model is zero (0) gauge pressure which is equal to the atmospheric pressure at the exit. However at the inlet down to the valve the maximum pressure difference is determined as 267.38Pa; bringing the absolute pressure (δp/δt) to 101592.38Pa ($101325_{pa} + \Delta P$). In a similar view, Prandtl mixing length hypothesis which was used for the calculation of the 2-dimensional model in figure 4 with the CFD code shows its effectiveness because the pressure difference was evaluated after 63-iterations with minimum and maximum difference of 0.816094Pa and 0.847877Pa respectively yielding a dynamic pressure of 0.031783Pa.

Figures 5 and 6 are result comparison using mathematical approach with the aid of the governing equations and the CFD simulation tool. The curves show close match. Thus results confirm the capability of CFD simulation with respect to real life scenario.

However, it is observed in figure 7 that much of the flow is from the inlet and it goes down the combustion chamber though a minimal percentage which is always around the valve stem. Similarly, flow observed from the outer diameter of the port is mainly responsible for the swirl inside the engine cylinder. Thus streamlines released from the inlet shows the flow feature in the figure. Swirl created is stronger and visible near top of the cylinder during lower lift but as the valve opens up, the swirl location moves down and gets weaker. This interpretation gives clear picture of the graphical illustration in figure 5.

VII. Conclusion

The test of flow process and charge motion on the Inlet Port of ICEs using a CFD code application was justifiable due to the following reason.

- Results certified that CFD calculations of the swirl ratio and flow coefficient are good enough to test for the fundamental equations of fluid dynamics.
- The CFD application tool is proficient enough to identify the region of the ICE port with more flow.
- CFD code determination of Prandtl mixing length hypothesis for dynamic pressure of fluid between two or more regions is effective.
- Dynamic pressure difference of 0.031783Pa across the model after 63-iterations was evaluated confirming fluid from one region to another within the inlet port.

Therefore, the study of CFD code application on flow process on the inlet port of internal combustion engines was feasible. However, the CFD approach on the research work was carried out without the consideration of shape optimisation; hence maximising the use of this feature of the tool will be advantageous for geometric optimisation.

Nomenclature

$\propto_k$	Flow coefficient
m_{theor}	Theoretical mass flow rate
m_{std}	Measured mass flow rate at standard condition
A	Piston Area
ρ_s	Density
C_s	Isentropic flow velocity
C_A	Cylinder Circumferential air speed
D_{cyl}	Cylinder bore diameter
K	Specific heat ratio (1.4)
n	Paddlewheelspeed (Swirl)
P_{amb}	Test ambient pressure
T_{amb}	Test ambient temperature
P_{std}	Standard ambient pressure
T_{std}	Standard ambient temperature
R	Universal gas constant
V	Volume flow rate
V_{real}	Theoretical volume flow rate
ρ_{real}	Density under test ambient conditions
ρ	Density under isentropic conditions
ρ_{std}	Density under standard ambient conditions

REFERENCES

[1] C. R. Kumar, and G. Nagarajan, "Investigation of flow during intake stroke of a Single cylinder Internal Combustion Engine," *ARPN Journal of Engineering and Applied Sciences*, Vol.7, No. 2, pp.180 –186, ISSN: 1819-6608, 2012.

[2] S. Sadakane, M. Sugiyama, H. Kishi, S. Abe, J. Harada, and T. Sonoda, "Development of a New V-6 High Performance Stoichiometric Gasoline Direct Injection Engine," *SAE paper* 2005-01-1152, 2005.

[3] J. B. Heywood, Internal Combustion Engine Fundamentals, USA: McGraw-Hill, 1988.

[4] S. K. Sabale, and S. B. Sanap, "Design and Analysis of Intake Port of Diesel Engine for Target Value of Swirl," *American Journal of Mechanical Engineering*, Vol.1, No.5, pp.138 –142, doi:10.12691/ajme-1-5-6, 2013.

[5] D. Nureddin, and Y. Nuri, "Numerical Simulation of Flow and Combustion in an Axisymmetric Internal Combustion Engine," *International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering*, Vol.1, No.12, pp.692–697, 2007.

[6] K. M. Pandey, and R. Bidesh, "CFD Analysis of Intake Valve for Port Petrol Injection SI Engine," *Global Journal of Researches in Engineering*, Vol.12, Issue 5, pp.13 –19, 2012.

[7] K. B. Murali, and J. M. Mallikarjuna, "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*, Vol.4, No.1, pp. 1-14, ISSN 1735-3645, 2011.

[8] B. Khalighi, "Study of the intake tumble motion by flow visualization and particle tracking velocimetry," *Journal of Experiments in Fluids*, Vol.10, pp.230-236, 1991.

[9] M. McCracken, and J. Abraham, "Swirl-Spray Interactions in a Diesel Engine," *SAE Technical Paper*, doi:10.4271/2001-01-0996, 2001.

[10] S. L.V. Prasad, V. Pandurangadu, B.V.V. Prathibha, and D. V. V. Naga, "Experimental study of the effect of air swirl in Intake manifold on diesel engine performance," *International Journal of Multidisciplinary Research & Advances in Engg. (IJMRAE)*, Vol.3, No.1, pp. 179 –186, ISSN 0975-7074, 2011.

[11] A. K. M. Mohiuddin, "Investigation of the Swirl Effect on Engine using Designed Swirl Adapter", *Iium Engineering Journal*, pp.197 – 205, 2011.

[12] Y. R. Pathak, K. D. Deore, and V. M. Patil, "In Cylinder Cold Flow CFD Simulation of IC Engine Using Hybrid Approach," *International Journal of Research in Engineering and Technology (IJRET)*, Vol. 3, Issue 8, pp. 16 –21, ISSN: 2319-1163, 2014.

[13] K. H. Y. Himanth, and N. Jayashankar, "Port Flow Simulation of an IC Engine", *International Journal of Innovations in Engineering Research and Technology (IJIERT)*, Vol. 2, Issue 9, pp. 1 – 9, ISSN: 2394-3696, 2015.

[14] H. Kim, S. Yoon, X. Xie, and M. Lai, "Effects of Injection Timings and Intake Port Flow Control on the In-Cylinder Wetted Fuel Footprints during PFI Engine Start up Process", *SAE paper*, 2005-01-2082, 2005.

[15] R. Abdul, R. K. Abdul-Razak, S. A. D. Mohammad, M. K. Ramis, "CFD Analysis of Flow Field Development in a Direct Injection Diesel Engine with Different Manifolds", *American Journal of Fluid Dynamics*, Vol.4, No.3, pp.102-113, doi: 10.5923/j.ajfd.20140403.03, 2014.

[16] R. D. Jebamani, N. M. T. Kumar, "Studies on variable swirl intake system for DI diesel engine using CFD", *Thermal Science*, Vol.12, No.1, pp. 25-32, doi:10.2298/TSCI0801025J, 2008.

[17] K. Newton, W. Steeds, T.K. Garrett, The Motor Vehicle, Butterworth Group, London: ISBN 10: 0408010827, ISBN 13: 9780408010825, 9th ed, 1989

[18] J. L. Lumley, Engines: An Introduction, Cambridge: University Press, ISBN: 0-521-64489-5, 1999.