JST: Engineering and Technology for Sustainable Development
Volume 33, Issue 3, July 2023, 001-009

Study on Effect of Piston Bowl Depth on Intake Air Flow Behaviour Using Ansys Fluent Cold Flow

Tran Dang Quoc
School of Transportation Engineering, Hanoi University of Science and Technology, Ha Noi, Vietnam
Corresponding author email: quoc.trandang@hust.edu.vn

Abstract

Gas flow behaviour is an extremely complicated question on internal combustion engine and plays a key role in air-fuel mixing performance. A proper mixture process significantly improves the combustion performance and thus enhances the engine efficiency. This study estimates the effect of different piston bowl depths on the flow field development inside the combustion chamber of a CNG engine using Ansys Fluent. The cold flow model was utilized without considering the combustion behaviour. The three-dimensional model was developed based on a single cylinder CNG converted engine. Four different bowl heights were applied while the compression ratio was maintained as the designed value of 10. The tumble and swirl as well as the turbulent flow pattern were investigated to consider the gas flow behaviour. The simulation results indicate that the tumble and swirl ratio increase with increasing of piston bowl depth. The case of bowl height 17 mm shows better flow behaviour and higher velocity for a good mixing performance as well as turbulence dissipation.

Keywords: Single cylinder CNG engine, piston bowl depth, tumble ratio, swirl ratio, turbulent kinetic energy (TKE).

1. Introduction

In recent decades, natural gas (NG) has been widely used as an alternative fuel for Internal Combustion Engine to reduce the dependence on petroleum fuels and pollutant emissions. Due to the chemical and physical properties, the combustion of natural gas dissipates a small amount of greenhouse emissions in comparison with gasoline and diesel [1]. Generally, natural gas contains primarily more than 90% of methane (CH4) and a trace amount of ethane, propane, nitrogen, carbon dioxide and water vapor [2]. To store and increase the volumetric energy of fuel for transportation applications, natural gas is compressed into a typical tank with a pressure of approximately 200 bar and so-called compressed natural gas (CNG) fuel [3].

CNG is considered a non-toxic and environment-friendly alternative fuel. Since CNG is a gaseous fuel, most existing CNG engine is dual fuel engines converted from petrol engines with minor modifications [4]. The CNG fuel has several advantages as an alternative fuel such as low price, low emission, non-toxic carbon hydroxide pollution and huge available quantity. However, as a newly developed system, the CNG engines have many drawbacks that can be overcome to optimize the engine performance. To enhance the thermal efficiency of natural gas, it is necessary to overcome two major problems which are low concentration and slow burning velocity [5]. Improving the burning velocity of CNG engines is quite difficult because the gas motion in the combustion chamber is complicated with the variation of kinetic energy continuous [6]. These variations are significantly influenced by the processes of intake and compression. In addition, the cross-section of the intake valve and geometric shape of the cylinder head or piston are major effective parameters [7].

Previous research has indicated that the effect of combustion chamber geometry on the motion of gas flows in cylinder was stronger compared to the gas intake parameters [8]. Saravanan Subramanian and co-workers [9] examined the influence of piston crown on swirl in the combustion chamber at the engine speed of 1500 rpm. The simulation results illustrated that the effect of piston bowl on turbulence kinetic energy is negligible due to similar compression ratio, but the best swirl ratio was obtained in the toroidal combustion chamber. S. K. Gugulothu and K. H. C. Reddy have carried out the CFD simulation in a two-valve four-stroke diesel engine to study the in-cylinder air behaviour with four different piston shapes such as flat piston, central bowl piston, offset bowl and inclined offset bowl piston. The results indicated that turbulent kinetic energy, length scale as well as swirl and tumble ratios in case of flat piston were higher than that of other cases [10]. On the other hand, on the same research conditions, the results of B. Harshavardhan claimed that the flat piston was more proper for spark ignition engine and the air - fuel mixture was more flammable [11]. Pargaonkar *et al.* [12] analysed the

ISSN 2734-9381
https://doi.org/10.51316/jst.167.etsd.2023.33.3.1
Received: December 7, 2022; accepted: March 28, 2023

effects of piston bowl geometry on combustion and emission characteristics in a direct injection diesel engine. The results illustrated that the motion and swirl in the cylinder were unstable due to poor velocity distribution in flat piston. However, the distribution of air and swirl formation could be enhanced by using piston bowl.

To develop CNG engine reaching requirements such as high thermal efficiency or high emission standards, this study analyses the flow pattern of intake air in a CNG converted engine using Ansys Fluent. Based on consideration in previous research, this paper focuses on analysing the effects of two parameters piston bowl depth (*Hb*) on the air motion in the cylinder to understand the behaviour of intake gas in the cylinder and contribute to the future research of CNG engine.

2. Methodology

2.1. Experiment

In this study, a computational fluid dynamics (CFD) model was developed based on an experimental CNG engine. To briefly study the effect of piston bowl geometry, the engine performance and combustion duration were measured with different piston bowl heights. A single cylinder diesel engine D15 was converted to fully supply CNG with the addition of an ignition system and CNG supply system. The ignition timing was adjusted to obtain the highest engine power. The compression ratio was maintained at 10:1 with a self-designed compression ratio variation system as a part of our work on QTC2015 engine. The experimental setup and engine information are detailed in our previous works [4]. Different piston bowl depths were conducted to consider the effects of piston geometry. Based on the engine performance results, the combustion chamber was numerically studied to analyse the intake air flow behaviours for a better understanding of the in-cylinder mixture formation. Fig. 1 shows the schematic of the combustion chamber.

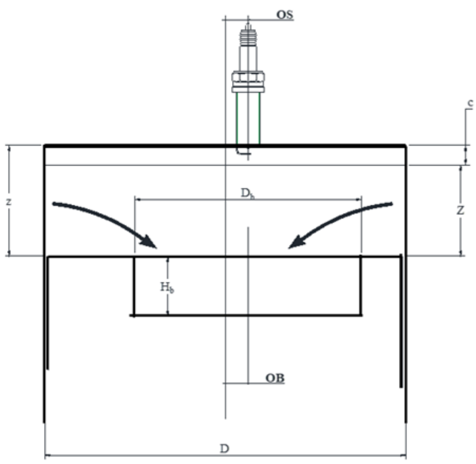

Fig. 1. Schematic of combustion chamber with piston bowl height *Hb*

The three-dimensional configuration model of combustion was developed based on the piston head geometries. The operating conditions were also utilized to verify the numerical model. The engine parameters are indicated in Table 1.

Table 1. Parameters of the CNG converted engine

Parameter	Symbol	Value	Unit
Cylinder bore	*D*	103	*mm*
Stroke	*S*	115	*mm*
Piston bowl diameter	*Db*	66	*mm*
Piston bowl depth	*Hb*	0, 5, 10 and 17	*mm*
Piston bowl offset	*Ob*	0	*mm*
Engine Speed	*n*	1800	*rpm*
Compression ratio	*ε*	10:1	-

The engine performances under different piston geometries were compared. The power and torque of the engine with different piston bowl depths are shown in Fig. 1. It can be seen clearly that the engine torque achieves a better performance with the piston bowl height of *Hb* = 17 mm. When the piston bowl depth increases from 0 to 17 mm, engine torque increases slightly to 47.35 Nm while the engine power increases significantly to 8.93 kW. However, the case of *Hb* = 25 mm obtains a small decline in engine performance. On the other hand, the knocking problem is observed when the engine is operated with the piston bowl depth of 25 mm. The bowl depth is also demonstrated to be limited with compression ratio due to knock limit [14]. Therefore, the simulation model *Hb* = 17 mm is the highest bowl heigh studied in this work. As mentioned before, the ignition timing was modified to obtain the best engine performance with every piston bowl instance. The optimized ignition timing with different conditions is shown in Fig. 2. Obviously, piston bowl configuration shows a strong impact to the CNG engine performance. Therefore, the combustion duration was estimated to further study the influence of bowl height on the fuel-burning rate.

The variation of combustion duration with different pistons is shown in Fig. 3. As estimated from the fuel burnt, the combustion duration is defined as the period from 10% to 90% of the mass fraction burnt. Noticeably, respectively to the engine performance, the piston bowl with a height of 17 mm achieves a quick combustion with a combustion duration of 65.9 degrees. A slightly increase in the case of *Hb* = 25 mm while a long combustion period is obtained with shallow piston bowls. Engine without piston bowl (*Hb* = 0 mm) has a very enduring combustion of 114.6 degrees which illustrates a slow flame propagation occurrence and a bad emission pollution.

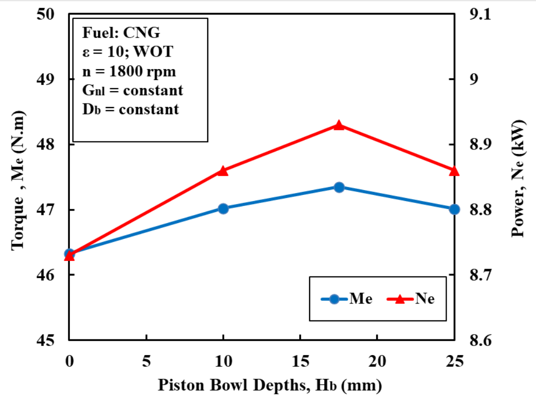

Fig. 2. Engine performance with different piston bowl depths.

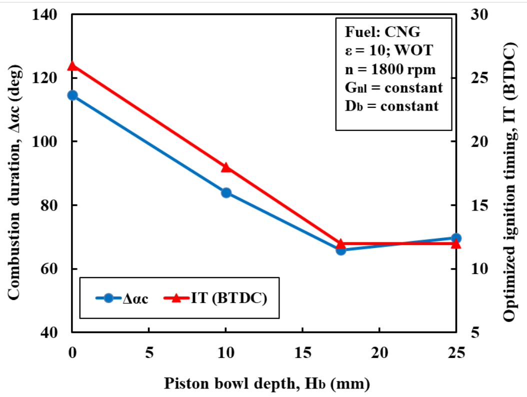

Fig. 3. Combustion duration and optimized ignition timing with different piston bowl depths

Therefore, the numerical model was studied to analyse the intake air flow behaviour of designed engine to have a better understanding of the impact of piston bowl depth on the engine performance and the fuel-air mixing process. The difference piston bowl depths of 0, 5, 10 and 17 mm is studied in the simulation analysis in this work.

2.2. Engine Modelling

Herein, Ansys Fluent software with internal combustion engine (ICE) Cold flow model is used to simulate and examine the motion of gas flows without chemical reaction in the combustion chamber. The turbulent flow parameters such as swirl ratio, tumble ratio, cross tumble ratio and mass-average turbulent kinetic energy are used to investigate the effect of piston bowl geometry on the in-cylinder gas behaviours. The 3D geometry models of different piston shapes with 4 different piston bowl depths Hb = 0, 5, 10 and 17 mm were built according to the experimental engine. The numerical analyses were considered during intake and compression stroke from the intake valve open time at CA = 350 deg to CA = 720 deg. The top dead centre (TDC) with valve overlap is considered as the position of CA = 540 deg. The engine parameter and simulation setup are shown in Table 1.

The input parameters of the cold flow model such as connecting rod length and crank radius were corrected by the experiment engine. The engine speed was constant at 1800 rpm for all cases while compression ratio was maintained at 10 regarding to the designed experimental engine. The intake flow was considered as wide-open throttle (WOT) for the full load operation. Since considering the transient gas flow, valve lift and valve timing are key factors to the flow behaviour over time. The valve lift profiles are obtained from the experimental engine as optimized in our previous work, as shown in Fig. 4. The moving mesh is applied to calculate the movements of the intake and exhaust valves as well as the piston head.

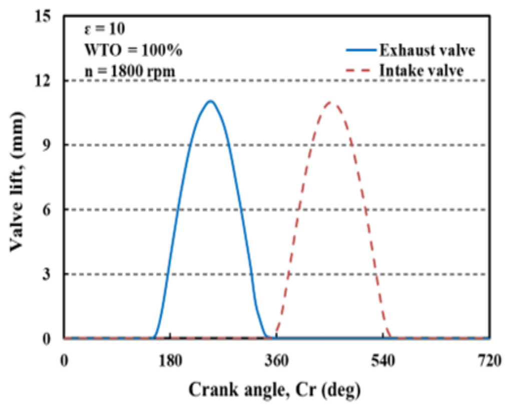

Fig. 4. Valve lift profiles using for simulation model

The governing equations such as the continuity equation, momentum equation and energy equation were solved in Ansys Fluent. Standard k - ε turbulent model was used to estimate the turbulence behaviour of the gas flow. In order to compute the numerical model and setting conditions for different positions in the engine, the control volume was decomposed into several zones. The control volume and simulation zones are indicated in Fig. 5. Different regions were indicated by different colours. The initial parameters of model including temperature, pressure as well as the position of valves and piston were input based on the experimental engine at the start of intake valve opening. Otherwise, the internal velocity of fluids is initially considered as zero. Turbulent kinetic energy is also negligible in the start of the simulation. The control volume is meshed based on the decomposed cell zones and interfaces.

Different zones of the engine and different positions in every zone have different quality and different shapes of the cell. The finer grid is, the more accuracy of the simulation can be obtained, but the more time for calculation is required. Therefore, optimizing the number of mesh cells is very important. The intake and exhaust valve seat zones require finer mesh cells to calculate the valve lift movement but, in the intake, and exhaust ports coarser mesh cells are acceptable. The interfaces also need finer mesh cells and require matching between two zones. Furthermore, lower zone of cylinder and piston bowl are made by hexahedral cell to simulate the movement of piston. Ansys workbench meshing was used for mesh generation. The number of cells was optimized to minimize the mesh dependence of model. The meshing of model is shown in Fig. 6.

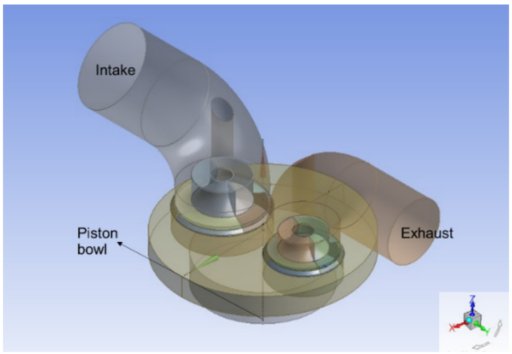

Fig. 5. Decomposition in the modeling geometry

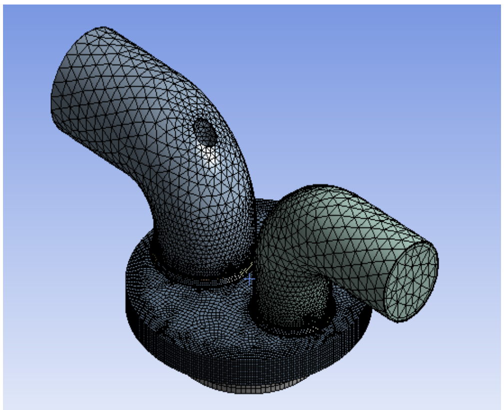

Fig. 6. Meshing of the cold flow model

To estimate the turbulence behaviour of in-cylinder flow, the specific velocity was evaluated. In addition, tumble ratio, swirl ratio and cross-tumble ratio are calculated in the CFD model. Tumble ratio (X-axis turbulence), cross tumble ratio (Y-axis turbulence) and swirl ratio (Z-axis turbulence) can be expressed respectively as flows:

$$TRx = \frac{w_x}{2\pi N} = \frac{60L_x}{2\pi I_x N} \quad (1)$$

$$TRy = \frac{w_y}{2\pi N} = \frac{60L_y}{2\pi I_y N} \quad (2)$$

$$SRz = \frac{w_z}{2\pi N} = \frac{60L_z}{2\pi I_z N} \quad (3)$$

where w_x, w_y and w_z are instantaneous velocity of fluid flows in the axis of x, y and z, respectively. N is the engine speed (rpm). L_x, L_y and L_z are angular momentums of fluid flow corresponding to x, y and z axis. I_x, I_y and I_z are inertial momentums of turbulent flows in x, y and z.

Since the direction of instantaneous velocities in (1), (2), (3) are considered, the positive values of ratios indicate the clockwise flows. In contrast, the counterclockwise flows are represented in the negative values of turbulent ratios. In order to reduce the computational effort, velocity contours were saved for every four CA degrees from 352 to 720 degrees. The k - ε turbulence model was used for turbulent flow estimation [13]. The turbulent kinetic energy (k) and turbulent dissipation rate (ε) are calculated in the numerical simulation.

3. Results and Discussions

3.1. In-Cylinder Flow Pattern

In the numerical model, geometries 4 different piston bowl depths Hb = 0, 5, 10 and 17 mm are considered to analyse the effect on the in-cylinder gas flow.

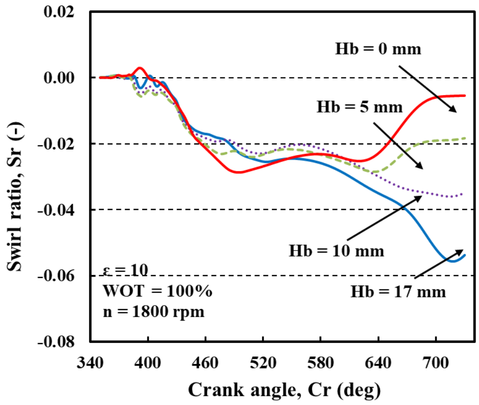

Fig. 7. Variation of Swirl ratio in cylinder according to crank angle

Fig. 7 shows swirl ratio in suction and compression stroke with different piston bowl depths Hb = 0, 5, 10 and 17 mm. Overall, deviation of swirl ratio generated by different piston bowl depths is trivial. Obviously, the depth of piston bowl does not affect the swirl eddies which have axis parallel to the cylinder centerline therefore its influence on swirl ratio is very small in this period. Swirl ratios in four cases only have different tendency from CA = 580 deg due to the compression. Magnitude of swirl ratio is higher with deeper piston bowl and the negative peak value of swirl ratio is obtained at TDC in the case of Hb = 17 mm. Swirl ratio in the case of Hb = 10 mm also has the peak value at TDC while the cases of Hb = 5 mm and Hb = 0 mm have the peak value at CA = 640 deg. In this duration, by the upward movement of piston, the in-cylinder gas is compressed into the bowl. As a result, the case of Hb = 17 mm shows a better swirl behaviour which is better for mixing performance. It can be explained that the deeper piston bowl, the bigger space for gas flow to develop. In the cylinder with shallower bowl, gas flow is compressed and decayed quickly in the squish area. This accounts for the deviation of swirl ratio in the late stage of compression stroke.

Variation of tumble ratio with different piston bowl depths during intake and compression stroke is shown in Fig. 8. It can be observed that piston bowl depth has strong effect on tumble ratio, especially in the intake stroke. At the beginning, tumble ratio has a small increase to the first maximum value at *CA* = 360 deg before quick decrease to negative value. From here, tumble ratio shows clear differences in 4 cases which represents the effect of piston bowl depth. The case of *Hb* = 0 mm has tumble ratio decreasing the most rapidly, the following are the cases of *Hb* = 5 mm, *Hb* = 10 mm and *Hb* = 17 m, respectively. All the cases achieve the negative value at *CA* = 406 deg in which the highest negative peak value obtained is -1.37 with *Hb* = 0 mm. In this stage, intake valve has wide open that accelerates the intake gas flow. The high velocity flow associating with downward movement of piston generates counter-clockwise tumble flow and gives tumble ratio negative value. On the other hand, the highest bowl depth obtained a higher positive peak at *CA* = 510 deg. Obviously, the piston bowl enhances tumble flow and changes flow direction. Furthermore, the tumble flow pattern is investigated at different position of piston.

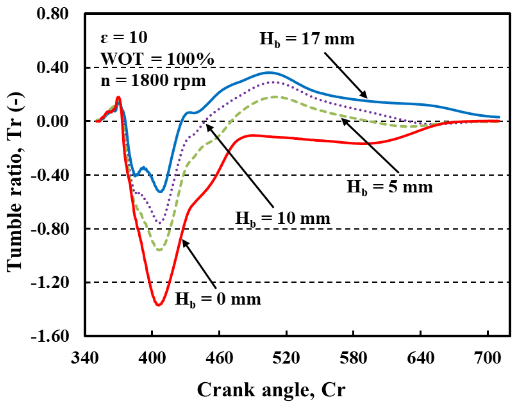

Fig. 8. Variation of tumble ratio in cylinder according to crank angle

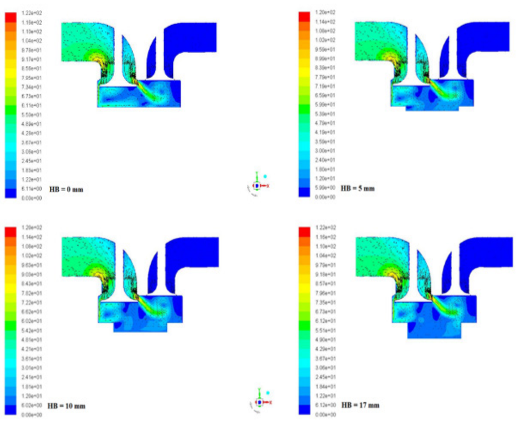

Fig. 9. Motion of gas flow into cylinder with different bowl depths (*CA* = 404 deg)

Fig. 9 indicates the behaviour of intake gas flow at *CA* = 404 deg with a peak of tumble ratio. It can be seen that a large eddy shows up in 4 cases. After reaching the negative peak at *CA* = 406 deg, tumble ratio increases, the in-cylinder gas spreads over the cylinder volume, turns toward the intake side and generates clockwise eddies. Tumble ratio increases to positive value and has the second maximum value at *CA* = 510 deg in the cases of *Hb* = 5, 10 and 17 mm. On the other hand, tumble ratio in the case of *Hb* = 0 mm increases slowly in negative area until the end of compression stroke. In here, by the compression and friction, tumble flow decays continuously into small eddies, tumble ratio approaches 0 in all the cases. It worth noting that large-scale eddies inhibit the mixing process and hence form heterogeneous mixture.

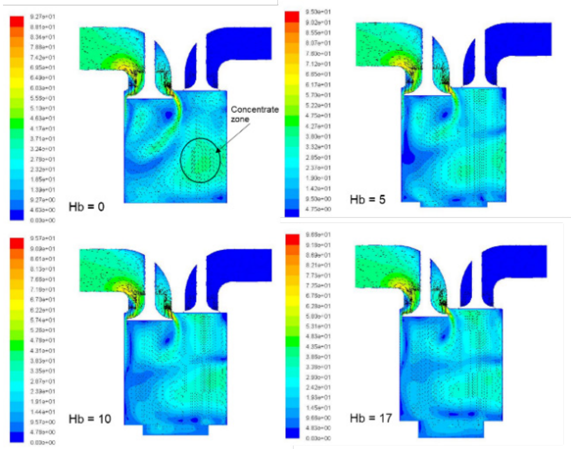

Fig. 10. Motion of gas flow into cylinder with different bowl depths (*CA* = 512 deg)

Fig. 10 shows the status of the in-cylinder air at *CA* = 510 deg. It can be seen that the large eddy is remained in the case of flat piston. A zone with concentrate fluid can be observed which account for a heterogeneous mixture. Meanwhile, a better dispensation is obtained when using a bowl piston. Piston with a bowl height of 17 mm achieves a well distribution throughout the combustion chamber. In addition, a slightly higher velocity can be observed when the piston bowl height increases.

Fig. 11 shows the velocity streamline of engine model in the case of *Hb* = 10 and 17 mm. The difference in two cases is small. However, it can be seen that the piston bowl with higher bowl depth has generated a higher velocity of fluid, especially near the intake valve zone. The higher velocity contributes to the mixing behaviour of fuel into air and perform a better combustion efficiency.

Fig. 12 shows cross tumble ratio during intake and compression stroke with different piston bowl depths. From this figure, it can be observed that cross tumble ratio has unclear deviations. This result proves that the effect of central bowl piston on the cross-tumble flow is small even though with different high of bowl. Cross tumble ratio only has a small-scale fluctuation in the compression stroke in the case of *Hb* = 17 mm which shows a small difference over the other cases. It decreases slightly and has the negative peak value at *CA* = 650 deg. And cross tumble ratios in the cases of *Hb* = 0, 5 and 10 mm have small fluctuations around -0.04 and 0.01. Obviously, the higher piston bowl depth still shows a better cross tumble flow of in-cylinder flow. However, the effect on cross tumble flow is very small compared to swirl ratio and tumble ratio.

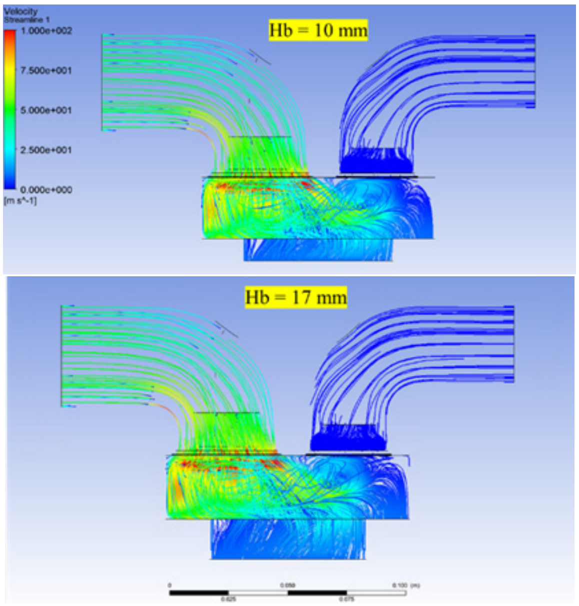

Fig. 11. Velocity streamline of engine model with different bowl depths (*CA* = 404 deg)

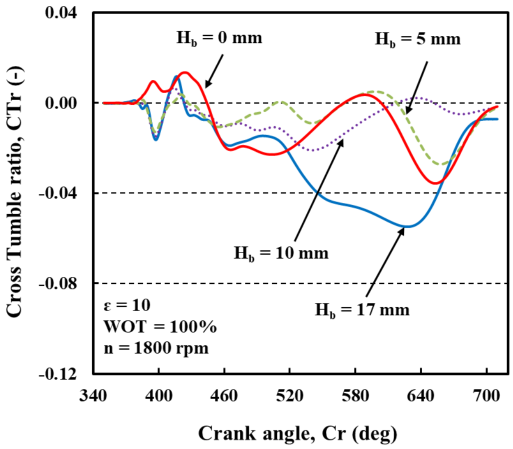

Fig. 12. Variation of cross tumble ratio in cylinder according to crank angle

3.2. Turbulence Flow Parameters

Moreover, the parameters of turbulence flow in the combustion chamber is estimated including turbulence intensity, turbulent kinetic energy and dissipation rate. The variation of turbulence intensity with different piston geometries is shown in Fig. 13.

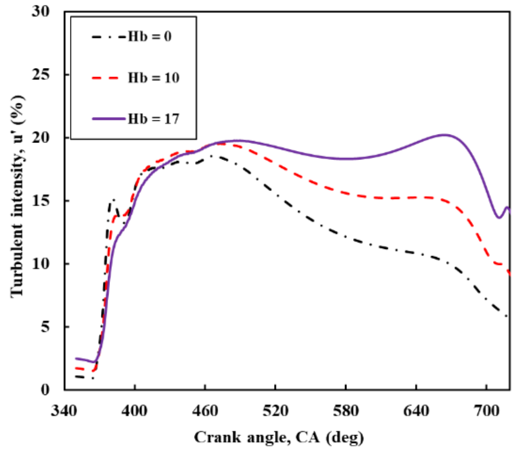

Fig. 13. Turbulence intensity of in-cylinder gas with different piston bowl depths

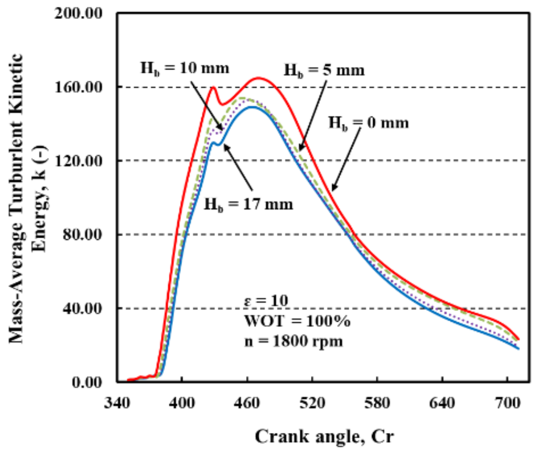

Fig. 14. Variation of mass-average turbulent kinetic energy in cylinder according to crank angle

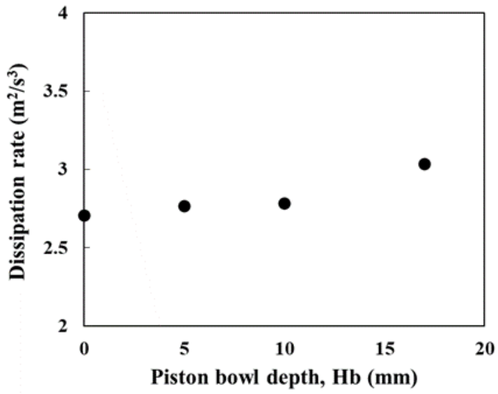

Fig. 15. Dissipation rate of different piston bowl depths

Turbulence intensity, referred to the turbulence level is defined as the ratio between the root-mean-square of turbulent velocity fluctuations and mean velocity. In other words, the higher turbulence intensity the more turbulent flow is. It can be seen that the turbulence intensity increase with increasing of piston bowl depth. The case of *Hb* = 17 mm maintains a high intensity of turbulence at approximately 20% from the intake stroke to the end of the compression stroke (from *CA* = 400 deg to *CA* = 700 deg). The high turbulence level in the compression stroke contributes to better fuel-air mixing when CNG is injected into the combustion chamber

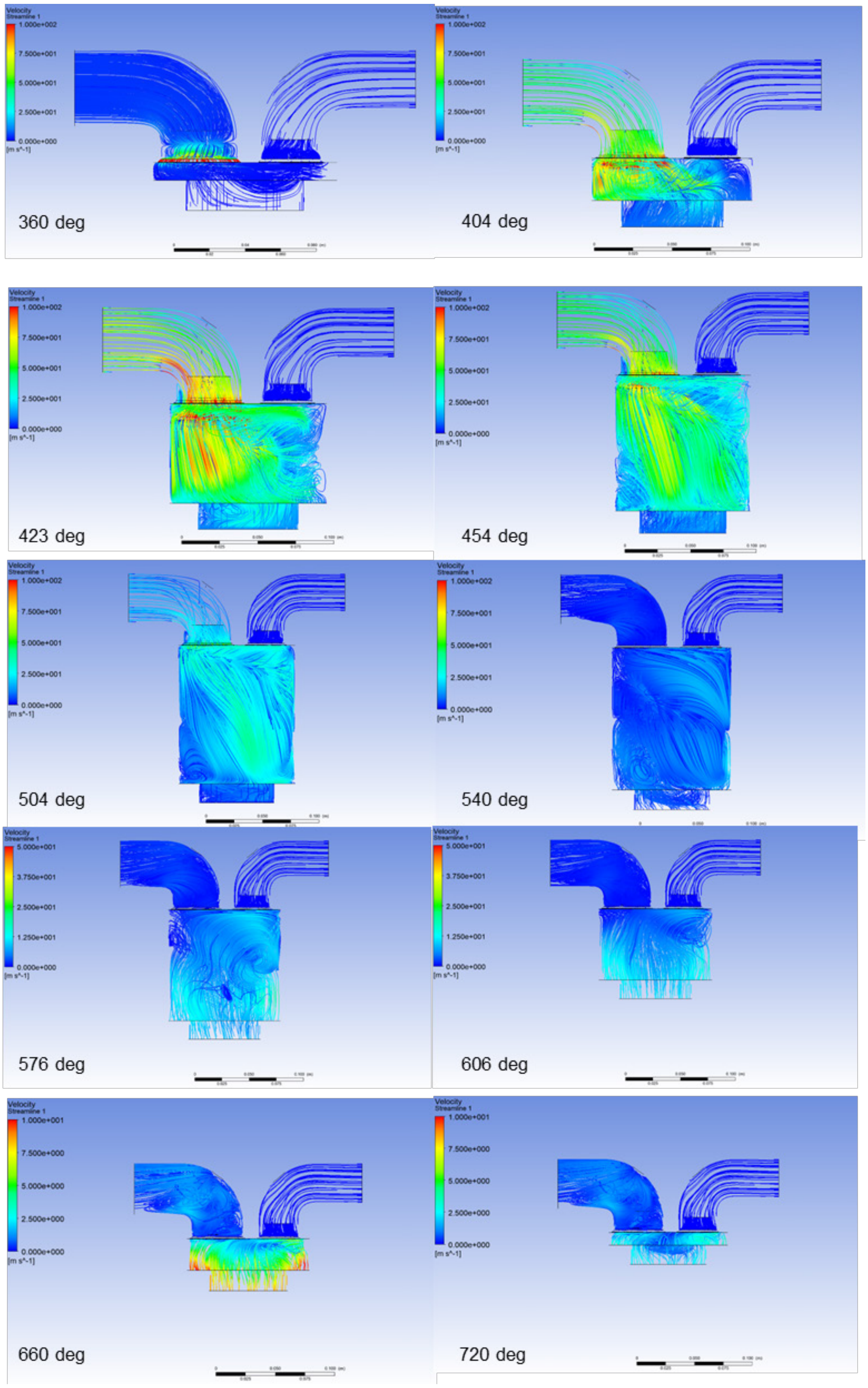

Fig. 16. Velocity streamline in different crank angle (*Hb* = 17 mm)

The mass-average turbulent kinetic energy of fluid flow in cylinder with different piston bowl depths is showed in Fig. 14. The figure shows that turbulent kinetic energy has significant differences in the magnitude but has the same fluctuation tendency. Mass-average turbulent kinetic energy increases rapidly in the intake stroke and achieves the maximum value at about *CA* = 460 deg to *CA* = 470 deg when the intake valve opens the most widely. It decreases deliberately after reaching the peak value. The case of *Hb* = 0 mm has higher value than the other cases and has the highest peak value at *CA* = 470 deg. From this observation, it is possible to say that piston bowl depth strongly affects on turbulent kinetic energy in the cylinder. This result illustrates that the turbulent kinetic energy in the case of low piston bowl depth remains in the large eddies with a small dissipation rate. Notably, small-scale turbulent flow is more effective in the mixing process. Therefore, the dissipation rate of turbulent kinetic energy is further investigated to confirm the dissipation of turbulent flows into small eddies. The turbulent dissipation rates with different piston bowl depths are indicated in Fig. 15. Obviously, the dissipation rate increases with increasing of piston bowl depths. The highest dissipation rate was obtained in the case of *Hb* = 17 mm at 3.03 m^2/s^3. As mentioned before, the higher dissipation rate illustrates the better turbulent flow with smaller eddies and thus promote the mixing at interface between fuel and air.

By the analyses above, it can be observed that piston bowl depth has a strong effect on tumble and turbulence. The case of *Hb* = 17 mm shows a better performance on turbulent flow coefficients. The quantity of small-scale turbulence flow has been improved with the increase of bowl depth. For furthur information, the development of in-cylinder air in the case of *Hb* = 17 mm is presented in Fig. 16.

5. Conclussion

Computational Fluid Dynamics in general and Ansys Fluent with Cold flow model in particular has so many benefits in researching and predicting the in-cylinder gas motion. In this study, the simulation results bring out a visualization about flow field development in the cylinder with different piston shapes. Some conclusions can be obtained as following:

- The piston bowl depth shows a strong effect on intake gas flow behavior.

- The gain in piston bowl depth increases the swirl ratio and cross tumble ratio. The highest values are obtained in the case of *Hb* = 17 mm.

- The negative tumble ratio is generated as intake valve open. However, the high piston bowl depth improves the clockwise tumble flows.

- The higher air velocity can be obtained with higher piston bowl depth which results in a better mixing performance.

- The high piston bowl depth shows a better performance of turbulent dissipate contribute to generation of small eddies.

These results and analyses are the basis to analyze the mixture preparation for the combustion process and contribute to optimize the design of piston bowl shapes for the experimental engine.

References

[1] R. Krishna, Conversion of diesel engine to CNG engine of commercial vehicles and emission control, in Proceedings of the 17th IRF International Conference, New Delhi, India, 2018, pp. 68-73.

[2] C. C. Chen, Y. L. Lu, G. Z. He, Estimating methane fugitive emissions from oil and natural gas systems in china, Huan Jing Ke Xue, 43 (2022) 4905-4913.

[3] Semin RA. A technical review of compressed natural gas as an alternative fuel for internal combustion engines. Am. J. Eng. Appl. Sci. 2008;1(4):302-11, https://doi.org/10.3844/ajeassp.2008.302.311

[4] Q. D. Tran, T. T. Tran, V. Nguyen Duy, An experimental investigation on performance of converted CNG engine by varying piston bowl geometry: A case study, J Air Waste Manag Assoc, 72 (2022) 361-369. https://doi.org/10.1080/10962247.2022.2028689

[5] Z. Zhao, X. Yu, Y. Huang, T. Sang, Z. Guo, Y. Du, L. Yu, D. Li, Study on combustion and emissions of a combined injection spark ignition engine with natural gas direct injection plus ethanol port injection under lean-burn conditions, ACS Omega, 7 (2022) 21901-21911. https://doi.org/10.1021/acsomega.2c02154

[6] M. Garg, R. Ravikrishna, In-cylinder flow and combustion modeling of a CNG-fuelled stratified charge engine, Applied Thermal Engineering, 149 (2019) 425-438. https://doi.org/10.1016/j.applthermaleng.2018.12.036

[7] W. Qin, M. Xie, M. Jia, T. Wang, D. Liu, Large eddy simulation of in-cylinder turbulent flows in a DISI gasoline engine, Applied Mathematical Modelling, 38 (2014) 5967-5985, https://doi.org/10.1016/j.apm.2014.05.004

[8] C. Garth, R.S. Laramee, X. Tricoche, J. Schneider, H. Hagen, Extraction and visualization of swirl and tumble motion from engine simulation data, Topology-based Methods in Visualization, Springer, 2007, pp. 121-135, https://doi.org/10.1007/978-3-540-70823-0_9

[9] S. Subramanian, B. Rathinam, J. I. J. Lalvani, K. Annamalai, Piston bowl optimization for single cylinder diesel engine using CFD, SAE Technical Paper, 2016, https://doi.org/10.4271/2016-28-0107

[10] S. Gugulothu, K. Reddy, CFD simulation of in-cylinder flow on different piston bowl geometries in a DI diesel engine, Journal of Applied Fluid Mechanics, 9 (2016) 1147-1155, https://doi.org/10.18869/acadpub.jafm.68.228.24397

[11] B. Harshavardhan, J. Mallikarjuna, CFD analysis of in-cylinder flow and air-fuel interaction on different combustion chamber geometry in DISI engine, International Journal on Theoretical and Applied Research in Mechanical Engineering (IJTARME), 2 (2013) 104-108.

[12] M. M. Pargaonkar, V. Jumbad, CFD analysis of piston bowl geometry for CI direct injection engine, Int. Res. J. Eng. Technol., 5 (2018) 567-572.

[13] Mohammadi, B. and Pironneau, O., 1993. Analysis of the k-epsilon turbulence model.

[14] S. V. Le, H. H. Chan, T. D. Quoc, A study on the effect of compression ratio and bowl-in-piston geometry on knock limit in port injection natural gas converted engine, The AUN/SEED-Net Joint Regional Conference in Transportation, Energy, and Mechanical Manufacturing Engineering, Springer Nature Singapore, Singapore, 2022, pp. 56-73. https://doi.org/10.1007/978-981-19-1968-8_6