INTERNATIONAL JOURNAL OF AUTOMOTIVE SCIENCE AND TECHNOLOGY

2022, VOL. 6, NO: 3, 233-241 www.ijastech.org e-ISSN: 2587-0963

Numerical and Experimental Investigation of the Flow Structure in a Diesel Engine with Different Piston Bowl Structure

İlker Temizer[1], Ömer Cihan[2*]

0000-0003-1170-3898, 0000-0001-8103-3063

[1]*Sivas Cumhuriyet University, Automotive Engineering Department, Sivas, 58140, Turkey.*
[2]*Hakkari University, Mechanical Engineering Department, Hakkari, 30000, Turkey.*

Abstract

In this study, unfired (cold) flow application was investigated in a single cylinder diesel engine with different combustion chamber geometries. In the experimental study, images obtained with the help of an endoscopic camera for different cycle points were instantly detected at constant speed. At the same time, velocity distributions of two different combustion chambers for different crank angles were analyzed in Ansys Forte software at before and after TDC. Thus, the flow distributions of different combustion chamber geometries in the chamber were compared. It can be said that regional swirl is formed in the newly developed combustion chamber geometry and develop in the chamber rather than the piston base compared to the standard combustion chamber. In addition, especially during the compression process, the squish movement of the bowl was observed with the movement of the piston. Here, it could be said that the newly developed chamber geometry is more effective than the standard bowl geometry. When the distribution of velocity vectors in the x-y and x-z axis were examined in numerical analysis, especially in the TDC position, it was determined that the interaction of the flow developed by the new bowl geometry with the fuel droplets was more evident. While it is seen that the vector velocity changes are close to each other at the 692^0 CA before the fuel injection, it is seen that higher flow velocities are formed for the standard combustion chamber at the 720^0 CA where the spraying continues and develops.

Keywords: Piston bowl structure, cold flow, swirl/squish moves, velocity distribution

Research Article

https://doi.org/10.30939/ijastech..1111073

Received 29.04.2022
Revised 29.06.2022
Accepted 04.07.2022

* Corresponding author
İlker TEMİZER
ilktemizer@gmail.com
Address: Sivas Cumhuriyet University, Manufacturing Engineering Department, Sivas, 58140, Turkey.
Tel: +903462191010-2312

1. Introduction

Today, many researchers are working to improve the performance and exhaust emissions of internal combustion engines. These days, when Euro 6 norms are applied for exhaust emissions in engines, it is of great importance to produce engines with low emission values. In-cylinder flow analysis is very important in determining these two important parameters. The here aim is to optimize the in-cylinder flow, increase the combustion efficiency and reduce the emission values [1].

Soruşbay et al. [2] modeled the cold flow and compared it with experimental data in a project study. The movement of gases entering the cylinder during the intake period continues after the closing of the intake valve and changed during the compression period. Especially in engines with a cylinder diameter of less than 80 mm, the tangential air velocity in the cylinder reached high values around the TDC (Top dead center). When the experimental data were examined, it was observed that the average tangential air velocities of this engine with a relatively large cylinder bore did not increase significantly towards the end of compression by moving in a turbulent manner.

The pocket geometry created on the pistons is an important criterion for cold flow tests and in the next stage engine combustion characteristics, engine performance and exhaust emissions. TKE, swirl ratio and tumble ratio parameters is very important in in-cylinder flows [3]. Azad et al. [4] looked at the cold flow behavior of a diesel engine at an engine speed of 2400 rpm. In conclusion, in-cylinder flow behavior for diesel engines; It has been observed that speed, temperature, pressure and turbulent kinetic energy are significantly affected. Wei et al. [5] created different cavity geometry on the piston in a direct injection diesel engine, and the swirl ratios of this geometry and the standard geometry were compared. It has been observed that the swirl ratio created by the new piston geometry in the cold flow analysis engine significantly reduces the exhaust emissions in AVL Fire software. The lowest exhaust emission values were taken at a swirl rate of 0.8.

Shafie and Said [6] studied cold flow analysis using various pocket type. Ansys software was used for analysis. More improved swirl and tumble were obtained from the modified piston with center pit in toroidal diameter, and flat piston has larger throat diameter. The piston model with toroidal bowl geometry showed a 34.8% improvement in swirl rate and 7% improvement in tumble rate compared to the original piston bowl geometry at the end of the compression stroke. In another study [7], in spark ignition engines, piston movement and the shape of the piston bowl affected the position of the tumble vortex. It has been observed that the tumble parameter significantly affects the compression process in the spark ignition engine. It has been noted that with a double-lobed piston bowl geometry according to the flat piston geometry, Swirl ratio increased by 66.67% the intake process and increased by 91.47% the compression process. Sushma and Jagadeesha [8] emphasized the effect of tumble and swirl ratios on piston bowl geometry. A four-stroke single-cylinder diesel engine was used in the numerical study. Analyzes were performed in Ansys Fluent software. A, B and C piston configurations were used. A higher swirl ratio and volumetric efficiency were obtained with B bowl geometry than A and C. On the other hand, higher TKE was obtained with C bowl geometry. Air movement in the cylinder has a great effect on energy efficiency. Raj et al. [9] studied air movement in a single-cylinder diesel engine. Four different bowl type (Center bowl on flat piston, bowl piston with curved offset, flat piston and inclined piston) were used. Higher TKE (about 27%), approximately 15% higher tumble ratio and higher turbulence density were obtained in the flat piston according to the other geometries.

Modeling varies according to engine types and different definitions are made [10], [11]. Some optical measurement methods are used to perform cold flow tests. Experimental methods such as Laser Doppler Velocity (LDV) and Particle Image Velocimetry (PIV) are the preferred optical measurement methods for in-cylinder flows [12]. Moreover, Magnetic Resonance Imaging is used in the health sector for imaging events inside body. The device can also be operated by engineers to see the mixture movements inside a closed volume [13]. Chemiluminescence is the method that visualizes the flow in the basic chemical composition of an absorbed liquid. The method not common. Snapshots are created with emitted light high-speed cameras, and these images are used in flow dynamics [14]. In addition, tests related to in-cylinder air movements, spraying process and flow can be performed using the Endoscope-Based Imaging system [15], [16].

Krishna and Mallikarjuna [17] applied to PIV technique that motored single-cylinder and an engine made of plexiglas material. It was run at engine speeds of 400, 600, 800 and 1000 rpm. It was concluded that the tumble speed and flow are independent of the engine and vary according to the crankshaft angle. Krishna et al. [18] used the PIV technique to see the in-cylinder variation of turbulent kinetic energy and air velocity. The air flow rate was measured with an orifice device. Engine speeds of 1000, 2000 and 3000 rpm were selected as the operating condition. Turbulent kinetic energy and air velocity (radially, axially) increase as soon as the intake valve is open and with engine speed.

Ayaz et al. [19] investigated the cold air movements inside chamber throughout intake and compression stroke of a diesel engine used in locomotives Analyzes were performed in Ansys Fluent software. Higher velocities were obtained in the transition zone between the pre-combustion and the main combustion chamber in the intake stroke compared to the other zones of the combustion chamber. It was also observed that the TKE weakened before the fuel was injected. As a result, it was concluded that the existing heavy-duty diesel engine combustion chamber was improved in terms of engine performance with the evaluations made on the turbulent kinetic energy (TKE) and swirl ratio.

In this study, in-cylinder flow analysis of a combustion chamber geometry developed for use in a diesel engine was investigated. The visual flow findings were compared in the standard combustion chamber engine and the new combustion chamber engine. The main purpose of this study is to reveal the changes in air movements caused by the geometry of the combustion chamber at the presence of visual findings. Mixture formation and in-cylinder air movements are very important in diesel engines in terms of low emission and high performance criteria. It is known that if the direction in the bowl geometry is not good, the air movements negatively affect the mixture formation and combustion events. For this reason, cold flow (non-combustion) analyzes of the new bowl geometry developed to increase in-cylinder axial, tangential and radial flow mobility and standard bowl geometry were investigated.

2. Material and Method

2.1 Experimental study

In-cylinder flow mobility in internal combustion engines is known as difficult and costly processes. Especially in the combustion process, the gas flow rate and temperatures make this situation even more difficult. Therefore, it is necessary to image the inside of the cylinder with the help of high-speed and resolution cameras and special test equipment. In this study, it is possible to instantly image the swirl, squish and turbulence events that may occur in the fluid, especially in cold flow conditions. First of all, a 5x3x3 cm glass consisting of plexiglass material was placed in this section by cutting the cylinder head in certain dimensions (a location where an image can be taken from inside the combustion chamber). Then, a channel was opened in the plexiglass glass to take images with a PCE VE 200 endoscopic camera (LED supported). This type of endoscopic camera is an industrial imaging device that is frequently used in engine and machine equipment. Fig. 1 shows the modified cylinder head.

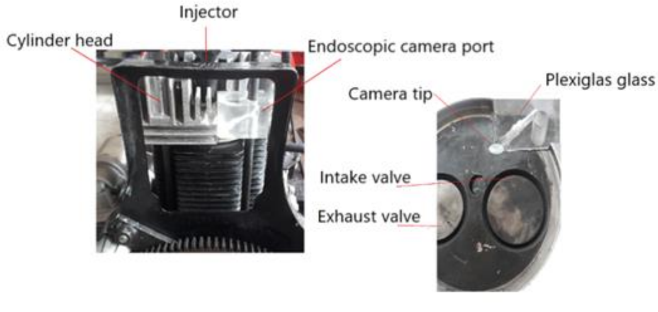


Fig.1. Modified cylinder head for cold flow testing

The experiments were carried out on Antor 3 LD 510 single-cylinder direct injection diesel engine, which was rotated at a constant speed (500 rpm) with the help of a dynamometer. The test setup of the engine is given in Fig. 2 and the technical properties of the engine for the test are shown in Table 1. In order to observe the flow mobility of the air in cold flow experiments, the air was represented by forming a smoke cloud. The air-gas mixture formed by the smoke bomb left in the air intake tank was sucked into the cylinder. The developed bowl geometry and standard piston were connected to the engine and cold operating tests were sequentially carried out. Moreover, the obtained results in the cold flow analyzes were compared with the numerical flow distribution.

Table 1. Technical properties of the tested engine

Engine Name	Antor 3 LD 510
Engine type	Four-stroke, air-cooled, single-cylinder direct injection diesel engine
Stroke volume	510 cm³
Stroke x Diameter	90 x 85 (mm x mm)
Compression ratio	17.5:1
Power	6.6@3000 (kW)
Torque	32.8@2000 (Nm)
Injection angle	126°
Number of injector holes	4

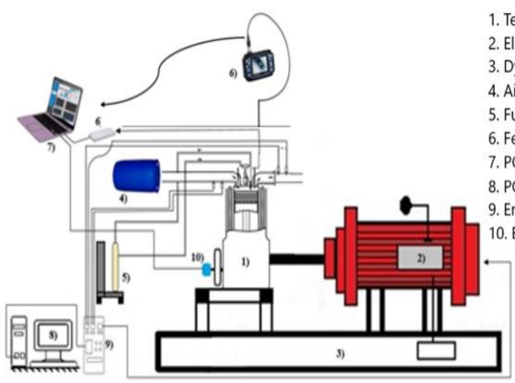


Fig. 2. Experimental setup

2.2 Numerical study

Based on the main dimensions and technical specifications of the combustion chamber of the ANTOR 3 LD 510 engine, the analysis of the existing combustion chamber geometry was carried out. In-cylinder heat, flow, mixture formation and combustion processes were performed in three-dimensional ANSYS FORTE software. Thus, the NCC model and the SCC model in the same chamber volume were examined. In the ANSYS program, time-dependent solutions were carried out in the turbulent flow regime to determine the velocity, pressure and temperature values. In the analysis of the designed bowl geometry were defined for the fuel-air mixture formation and combustion processes. This bowl geometry was chosen because it provides the engine's compression ratio of 17.5. The boundary conditions used in the numerical studies are given in Table 2.

Table 2. Simulation boundary conditions

Number of cylinders	Units	Single cylinder
Type of cooling		Air-cooled
Bore	mm	85
Compression ratio		17.5
Crank radius	mm	42
Number of injection nozzl		4
Injection Timing (start an	CA	705 and 729
Injection spray angle	deg	160
Injection rate (mass)	kg	5.11e-6
Engine speed	rpm	2000
Air inlet temperature	K	293.15
Air inlet pressure	bar	1
Fuel injection temperature	K	330.15
Turbulence model		RNG k-epsilon
Wall interaction model		Walljet1
Evaporation model		Multi component
Soot emission model		Kinetic model
NO emission model		Zeldovich model
Cylinder head temperature	K	575.15
Cylinder wall temperature	K	475.15

During the compression process in internal combustion engines, the air flow movement in the chamber decreases depending on the piston moving from BDC to TDC. This situation also has a significant effect on the mixture formation. In the three-dimensional CFD analysis, the full model is solved. In order to obtain more realistic exhaust emission values, a detailed reduced chemical combustion mechanism has been used, the accuracy of which has been tested in many studies. RNG k-epsilon turbulence model was used which is more suitable as sub-models in the study. This model offers the opportunity to examine in a macro dimension and provides an advantage over other models in terms of solution time. It is currently the most preferred model for the engine model in the literature [20, 21]. Adaptive Collision Mesh model is used for droplet collision. One of the biggest advantages of this model is that it eliminates the dependency on the network structure. The KH-RT hybrid model, including the Gas-Jet sub-model was used to break up the droplets. The KIVA-based wall collision model developed by Han was used to calculate the droplets colliding with the wall. The mesh structures of the new bowl geometry and the standard bowl geometry are given in Fig. 3. Most of the equations used in numerical analysis are given below. The continuity equation for Turbulent Reactive Flows is as follows (Equation 1).

$$\frac{\partial \rho}{\partial t} + \nabla . (\rho \ \tilde{u}) = \dot{\rho}^{s} \quad (1)$$

The momentum equation for the fluid (Equation 2) in addition to parameters such as convection, turbulent convection, pressure force and viscous tension, the effect of mass force and liquid sprays are calculated.

$$\frac{\partial \rho \tilde{u}}{\partial t} + \nabla.(\rho \tilde{u}\tilde{u}) = -\nabla \rho + \nabla \sigma - \nabla.\Gamma + \bar{F}^s + \rho \ \bar{g} \qquad (2)$$

Where, p is the pressure, Fs is momentum gain rate per unit volume because of spray, g is the specific mass force, σ is the viscous stress tensor given below (Equation 3).

$$\sigma = \rho \ \upsilon[\ \nabla\tilde{u} + \ \nabla\tilde{u}^T - \frac{2}{3}(\nabla\tilde{u})I] \qquad (3)$$

I is the descriptive tensor and υ shows the laminar kinematic viscosity. T is the subindex tensor transpose. Stress; Γ indicates the effects of non-linear ensemble mean transport nomial. On RANS approximation this is known as the Reynolds strain, on LES approximation it is defined as the SGS strain.

According to the First Law of Thermodynamics, the change of internal energy must be balanced through heat transfer and pressure work. Mixture issues with internal combustion engines; Convection of a multicomponent flow must be calculated taking into account the effects of turbulent attenuation, enthalpy diffusion, turbulent transport, sprays and chemical reactions. Internal energy transport equation (Equation 4):

$$\frac{\partial \rho \ I}{\partial t} + \nabla.(\rho \ \tilde{u}I) = \rho\nabla.\tilde{u} - \nabla.\bar{J} - \nabla.H + \rho\varepsilon + \dot{Q}^C + \dot{Q}^S \qquad (4)$$

J is the heat flux, and I is the specific internal energy (Equation 5,It occurs by enthalpy diffusion and heat conduction),

$$\bar{J} = \ \lambda \nabla T - \rho D \sum_k h_k \nabla \ y_k \qquad (5)$$

λ, T, hk correspond to the heat transfer coefficient, flow temperature and specific enthalpy expressions of the species, respectively. ε are turbulent kinetic energy dissipation, and Q^c and Q^s are chemical heat dissipation and spray interaction coefficients, respectively. The H term indicates the effect of the convection term on the ensemble mean.

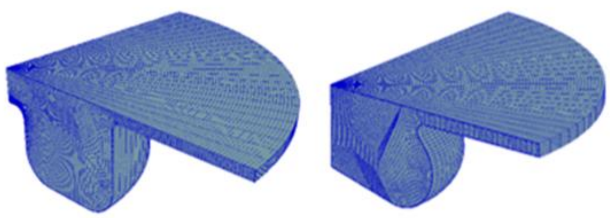


Fig.3. Mesh structure of the standard combustion chamber (left) and the combustion chamber with the new bowl geometry (right)

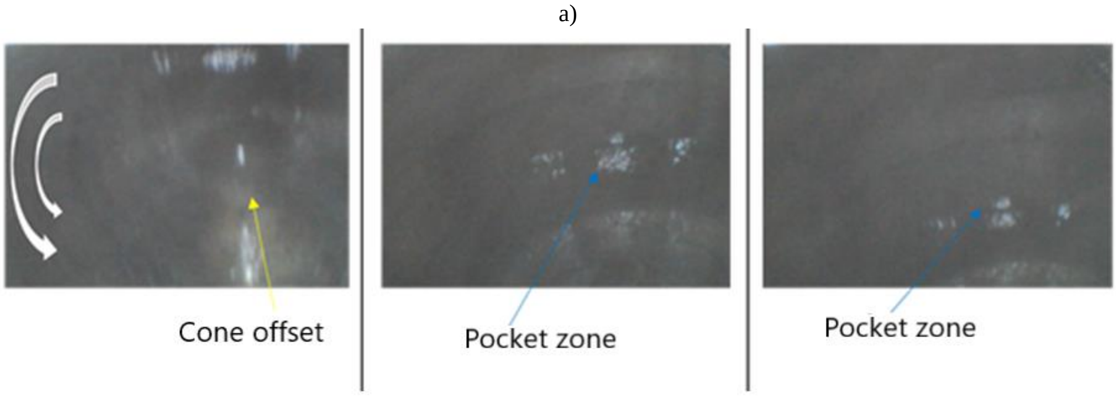
3. Result and Discussion

The in-cylinder air flow characteristics of the diesel engine using two different combustion chamber bowl geometries were investigated in the cold flow experiments. Changes in the smoke density of the gas taken into the cylinder determined the flow structure. Tests were repeated many times to obtain images in the engine, in which two different types of the bowl geometries were used. Snapshots taken from different points at the beginning of the intake time in the engine with the new bowl geometry are shown in Figure 5a and Figure 5b, the images of the end of the compression process in Figure 5c, the initial stages of the expansion process in Figure 5d and the flow images of the exhaust process are shown in Figure 5e. The snapshots of the standard combustion chamber obtained at different times are given in Fig. 5. Figure 5a images show the snapshots towards the beginning and end of the intake time, Figure 5b images show the compression and expansion start processes, and Figure 5c show the flow images of the exhaust times.

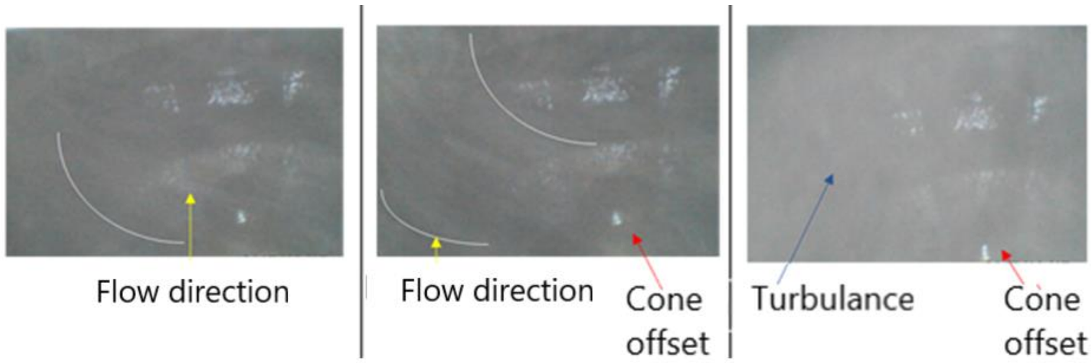
Since there is no scaled quantity obtained in numerical analyzes for cold flow experiments, the comments were made by looking at the air movements. Swirl, squish and turbulent movements on the piston were detected for both combustion chamber geometries in the cold flow analyzes for four stroke in the engine. With the opening of the intake valve, the air hitting the cavity area in the engine with the new bowl geometry was directed towards the center of the combustion chamber. It can be said that the increasing axial and tangential mobility of the fluid causes turbulence during the compression process, which starts with the movement of the piston towards the TDC (Fig. 4c). At this point, a trend similar to the compression process that takes place in standard bowl geometry can be mentioned (Fig. 5b). When Figure 5b is examined, it can be said that axial movements occur more intensely in the wall region. In this case, it has been observed that the axial movements of the piston with the new bowl geometry are intertwined with the radial movements. In this respect, the mixing process of the fuel sprayed into the combustion chamber with air was better in the new bowl geometry than in the standard geometry. When the exhaust processes are examined, it can be said that the tangential forces formed in the swirl turn into a vortex in a region (Fig. 4e). This situation was also observed in the standard bowl geometry (Fig. 5c). However, it can be said that while it occurs locally in the new bowl geometry, it occurs more along the piston in the standard geometry.


Piston head
Cone offset
Pocket zone

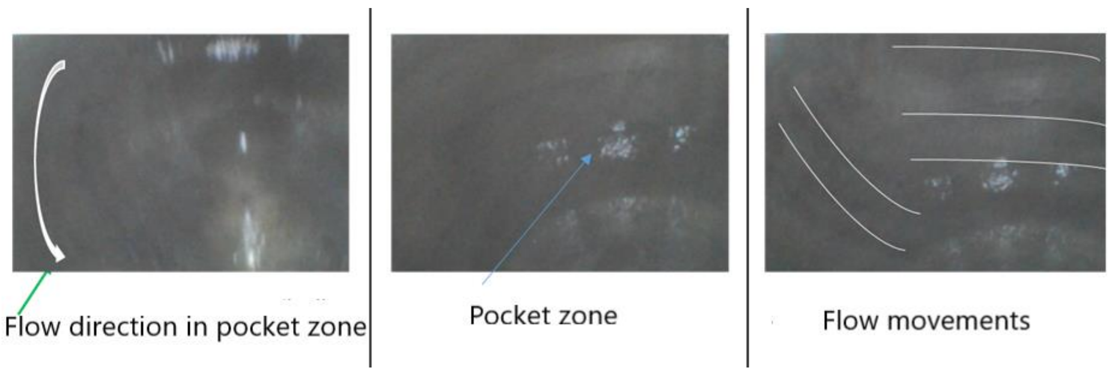
a)


Cone offset
Pocket zone
Pocket zone

b)


Flow direction
Flow direction
Cone offset
Turbulance
Cone offset

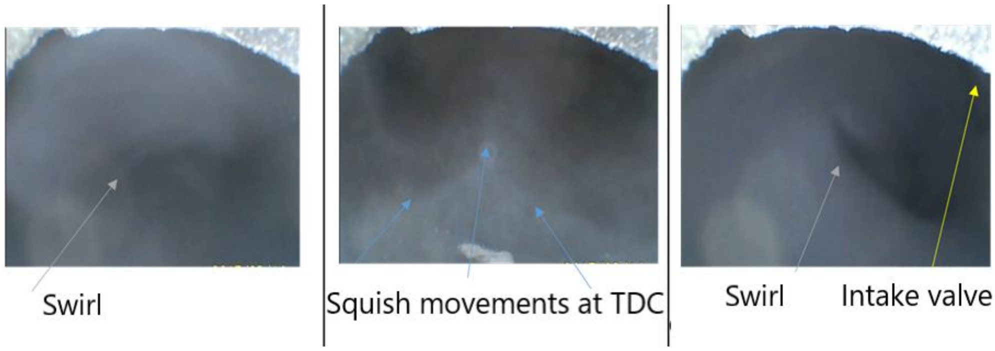
c)


Flow direction in pocket zone
Pocket zone
Flow movements

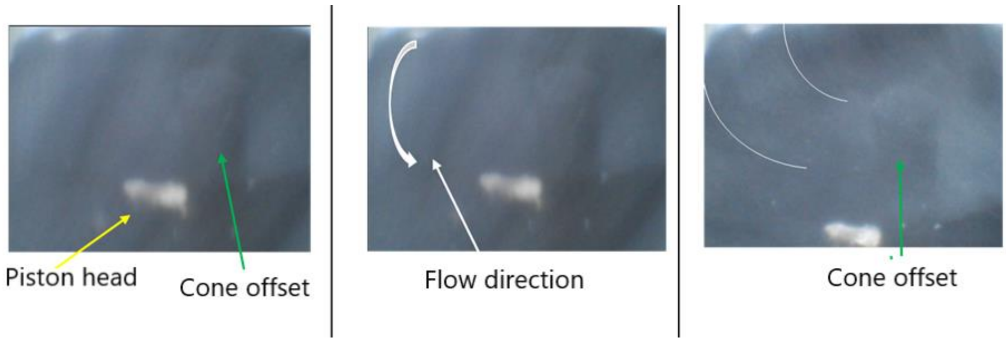
d)

e)

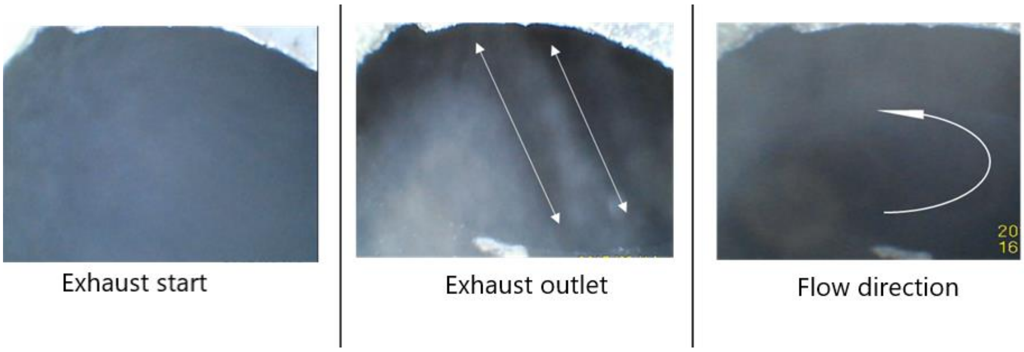
Fig. 4. In-cylinder cold flow images of the piston with the new bowl geometry at different times (a, b, c, d and e)



a)



b)



c)

Fig. 5. In-cylinder cold flow images of the piston with the new bowl geometry at different times (a, b and c)

Since there is no quantitative size in cold flow experiments, the comments should be supported by numerical analysis results. Experimentally, although fuel injection did not occur in cold flow analyses, pre-injection and post-injection analyzes were also analyzed in numerical analyses. Thus, in the next step, preliminary information about the distribution of the fuel assembly is provided.

Swirl is called the rotational movement of air around the cylinder axis and is one of the important air movements in diesel engines. The mixture of the injected fuel in the air with the movement of the piston to the TDC can be considered as an important parameter. In the simulations, the swirl ratio is assumed to be equal for both bowl geometries with the effect of the angular momentum or the initial rotational motion of the air with the opening of the intake valve. However, it can be said that this situation changes for different combustion chambers. Figure 6-8 shows varying velocity vectors for different crankshaft angles in the x-y plane. It can be said that the variation of the swirl ratios develops in parallel with the vector velocity distributions. While it is seen that the vector velocity changes are close to each other at 692° CA before spraying, it is seen that higher flow rates occur for the standard geometry at 720° CA where the spraying continues and develops. It can be said that these eddy movements and speed changes developing in the axial direction cause the fuel to more scatter in standard geometry. Especially in high speed engines, this results in a loss of inertia of the fuel. On the other hand, it can be said that the fuel-wall interaction develops on a larger surface in the new bowl geometry.

Velocity changes in the x-z plane of two different combustion chambers are examined in Figure 9-11. Especially in the new geometry, it can be said that the diameter in the throat part of the bowl expands a little and therefore the speed decreases a little in this region. When the new bowl geometry is compared to the standard geometry, the fuel directing function and the air-fuel mobility brought by the curved geometry are more realized on the radius that joins towards the base. This situation was also observed in cold flow analysis. Therefore, it can be said that the squish mobility in the new bowl geometry occurs more than in the standard geometry. As a matter of fact, the squish movement is known as the radial inward movement of the compressed air exiting the area between the top of the piston and the cylinder head as the piston approaches the TDC at the compression stroke. As the air is entrained and accelerated with the movement of the piston towards the TDC, the fuel-air mixture formation injected into the combustion chamber is improved. This process continues until the fuel droplets mix with the air and evaporate. In this respect, it is possible to use the new bowl geometry in terms of wall-fuel interaction.

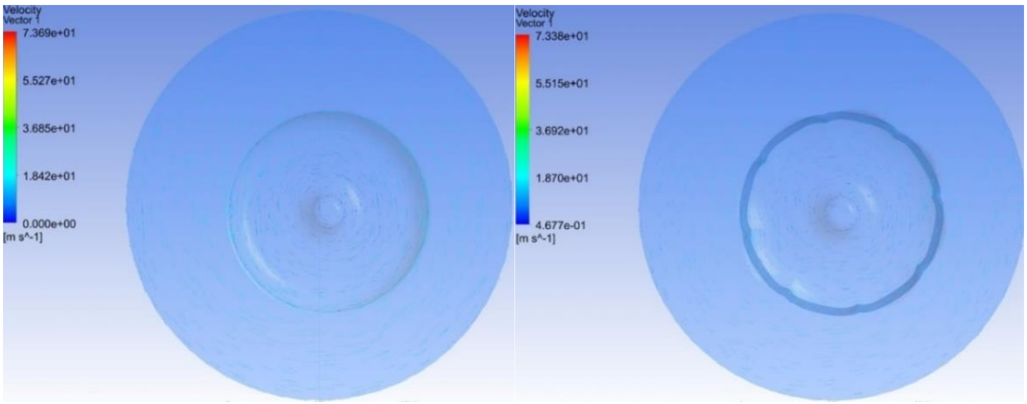


Fig. 6. Fluid velocity distribution in the x-y plane before TDC (692° CA)

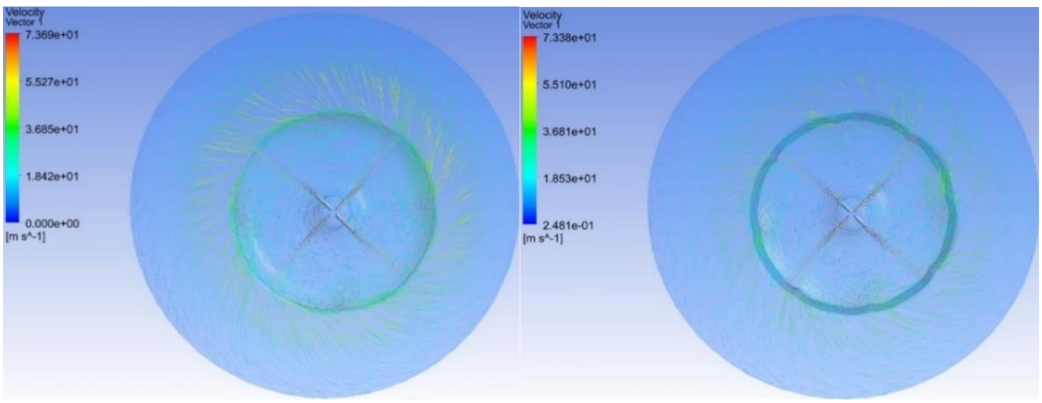


Fig. 7. Fluid velocity distribution in the x-y plane in TDC (720° CA)

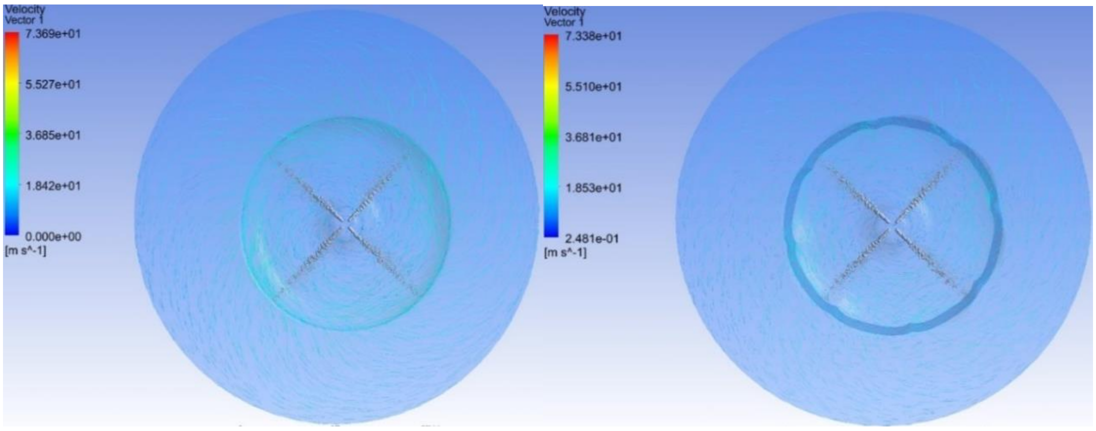


Fig. 8. Fluid velocity distribution in the x-y plane after TDC (730° CA)

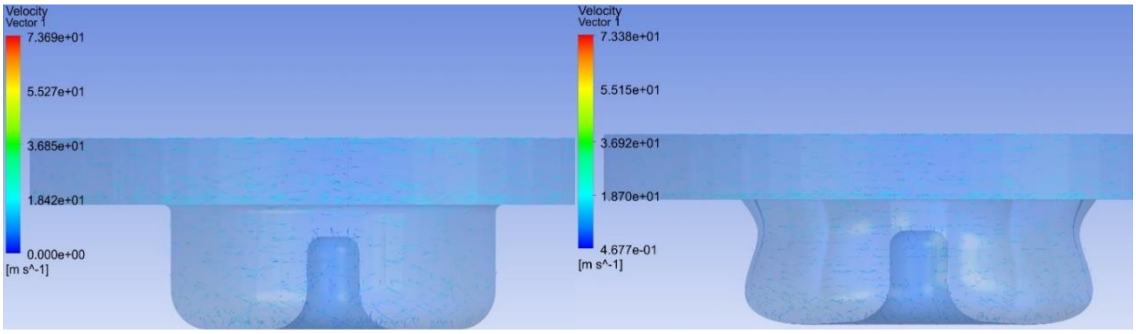


Fig. 9. Fluid velocity distribution in the x-z plane before TDC (692° CA)

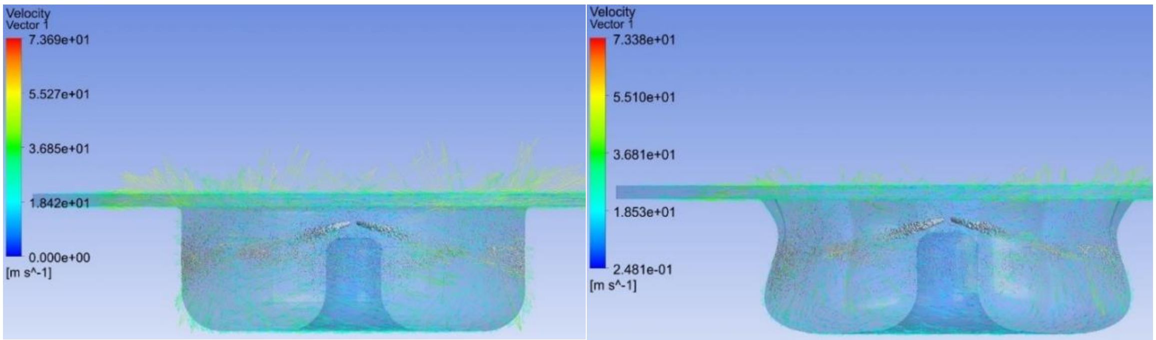


Fig. 10. Fluid velocity distribution in the x-z plane in TDC (720° CA)

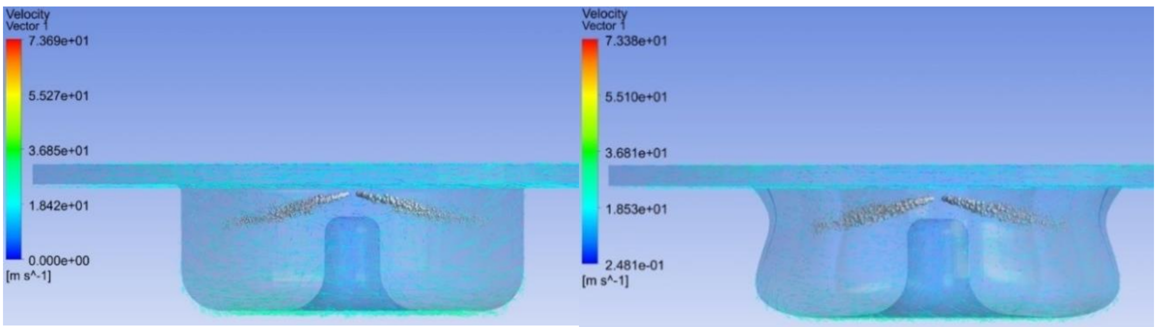


Fig. 11. Fluid velocity distribution in the x-z plane after TDC (730° CA)

4. Conclusion

The non-combustion (cold) flow motions of the new bowl and standard bowl geometries were experimentally investigated, and then the air-fuel interaction was numerically investigated at three different crank angles (692°, 720° and 730° CA). The obtained findings are listed in the following items.

-It can be said that the axial mobility in the standard bowl geometry is more limited compared to the new bowl geometry.

-Especially in the new bowl geometry, it has been observed that the flow movements developing in the axial direction (in the z direction) turn into radial (in the x direction) movements.

-It has been determined that swirl movements are concentrated at the piston base in the standard geometry, but this situation is limited in the new bowl geometry.

An increase in surface area in the combustion chamber causes to the increase of flow resistance. This increase limited the flow movements in the new combustion chamber. However, increases in surface area could be directly affect fuel pyrolysis in engines and accompanying exhaust emissions.

-Especially in numerical analysis, it was observed that radial and axial air mobility affect the distribution of fuel droplets.

-It is thought that the fuel-air interaction will have a positive effect on combustion and exhaust.

Acknowledgment

This research was supported by the Scientific and Technological Research Council of Turkey –TUBITAK Project No: 120M143.

Conflict of Interest Statement

The authors declare that there is no conflict of interest in the study.

CRediT Author Statement

İlker Temizer: Conceptualization, Supervision, Writing-original draft, Validation,
Ömer Cihan: Conceptualization, Writing-original draft, Validation, Data curation, Formal analysis

References

[1] J. B. Heywood, Internal Combustion Engine. Fundamentals. 1988.
[2] C. Soruşbay, M. Ergeneman, A. İmren, and A. T. Çalık, 'Experimental and Numerical Air Flow and Spray Dynamics Investigation in Internal Combustion Engines', Istanbul, 2008.
[3] A. Jamil, M. B. Baharom, and A. R. A. Aziz, 'IC engine in-cylinder cold-flow analysis – A critical review', Alexandria Eng. J. 2021; 60(3): 2921–2945.
[4] A. K. Azad, P. Halder, K. Nanthagopal, and B. Ashok, Investigation of diesel engine in cylinder flow phenomena using CFD cold flow simulation. Elsevier Ltd, 2019.
[5] S. Wei, F. Wang, X. Leng, X. Liu, and K. Ji, 'Numerical analysis on the effect of swirl ratios on swirl chamber combustion system of di diesel engines', Energy Convers. Manag. 2013;75:184–190.
[6] N. A. M. Shafie and M. F. M. Said, 'Cold flow analysis on internal combustion engine with different piston bowl configurations', J. Eng. Sci. Technol. 2017;12(4):1048–1066.
[7] B. Bibu and V. Rajan, Numerical simulation of cold flow analysis of internal combustion engine with double-lobed piston head. Springer Singapore, 2019.
[8] H. Sushma and K. B. Jagadeesha, 'CFD modeling of the in-cylinder flow in direct-injection Diesel engines', Int. J. Sci. Res. Publ. 2013; vol. 3, no. 12, pp. 995–1021. doi: 10.1016/j.compfluid.2003.09.003.
[9] A. R. Gnana Sagaya Raj, J. M. Mallikarjuna, and V. Ganesan, 'Energy efficient piston configuration for effective air motion - A CFD study', Appl. Energy, vol. 102, pp. 347–354, 2013,
[10]B. Somerville, 'A study of air motion and combustion in the DI Diesel engine', 1993.
[11]G. R. French and W. M. Scott, 'Giving the IDI diesel a fresh start', SAE Tech. Pap., vol. 850452, pp. 1–12, 1985, doi: 10.4271/850452.
[12]J. Rabault, J. A. Vernet, B. Lindgren, and P. H. Alfredsson, 'A study using PIV of the intake flow in a diesel engine cylinder', Int. J. Heat Fluid Flow. 2016; 62:1–17.
[13]P. Janas, I. Wlokas, B. Böhm, and A. Kempf, 'On the evolution of the flow field in a spark ignition engine', Flow, Turbul. Combust., vol. 98, no. 1, pp. 237–264, 2017, doi: 10.1007/s10494-016-9744-3.
[14]V. Granet, O. Vermorel, C. Lacour, B. Enaux, V. Dugué, and T. Poinsot, 'Large-Eddy simulation and experimental study of cycle-to-cycle variations of stable and unstable operating points in a spark ignition engine', Combust. Flame. 2012;159(4): 1562–1575.
[15]M. Bakenhus and R. D. Reitz, 'Two-color combustion visualization of single and split injections in a single-cylinder heavy-duty D.I. diesel engine using an endoscope-based imaging system', SAE Tech. Pap., vol. 1999-01–11, pp. 1–18, 1999.
[16]A. Manoj Babu, C. G. Saravanan, M. Vikneswaran, V. Edwin Jeo, and J. Sasikala, 'Visualization of in-cylinder combustion using endoscope in spark ignition engine fueled with pine oil blended gasoline', Fuel, vol. 263, pp. 1–9, 2020, doi: 10.1016/j.fuel.2019.116707.
[17]B. M. Krishna 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', J. Appl. Fluid Mech. 2011;4(1)1–14. doi: 10.36884/jafm.4.01.11895.
[18]B. Murali Krishna, A. Bijucherian, and J. M. Mallikarjuna, 'Effect of intake manifold inclination on intake valve flow characteristics of a single cylinder engine using particle image velocimetry', World Acad. Sci. Eng. Technol., vol. 68, pp. 176–182, 2010.
[19]E. Ayaz, H. Köten, and S. Çadırcı, 'Ağır iş dizel motoru silindiri içerisindeki soğuk hava akışının sayısal incelenmesi', Marmara Fen Bilim. Derg., vol. 4, pp. 445–452, 2018.
[20]D. Liu, X. Li, H. Shang, L. Xie, Y. Chen, and J. Chang, 'Combustion performance and fuel injection timing adaptability of a lateral swirl combustion system for direct injection diesel engines', Fuel, vol. 286, pp. 1–15, 2021, doi: 10.1016/j.fuel.2021.120663.
[21]İ. Temizer, Ö. Cihan, B. Eskici, 'Numerical and experimental investigation of the effect of biodiesel/diesel fuel on combustion characteristics in CI engine', Fuel. 2020;270:1-9.