CFD ANALYSIS OF THE IMPACT OF TWIN BOWL PISTON SHAPE ON IN-CYLINDER FLOW AND COMBUSTION IN A DI DIESEL ENGINE

Mahmut KAPLAN [1] Mustafa OZBEY [2]

[1] Amasya University, Technology Faculty, Mechanical Engineering Department, Turkey
*e-mail of the corresponding author: mahmut.kaplan@amasya.edu.tr
[2] Ondokuz Mayıs University Engineering Faculty, Mechanical Engineering Department, Samsun, Turkey

Abstract. *In this study, TB1 and TB2 twin bowl piston configurations in a direct injection (DI) diesel engine are produced by placing these configurations along the x-axis and y-axis, respectively, using SOLIDWORKS. The influences of the configurations on enhancing in-cylinder air motion and combustion parameters are examined using RNG k-ε turbulence model and the species transport model coupled with the eddy-dissipation model in ANSYS FLUENT. The simulation results show that both TB1 and TB2 configurations improve the organized rotating flows. The maximum swirl ratio of 1.03 and the maximum tumble ratio of 0.68 are obtained by TB1 and TB2 configurations, respectively, near TDC. Therefore, TB1 and TB2 configurations augment the indicated fuel conversion efficiency by 1.16 and 2.83%, respectively, compared to the base model. The major contribution of the present work to literature is to improve understanding details of in-cylinder fluid motion and help experimental studies related to enhancing diesel engine performance by using CFD model.*

Keywords: *DI diesel engine; CFD; Piston shape modification; Swirl and tumble flows;Combustion characteristics*

1. Introduction

Homogenous air-fuel mixture and rapid vaporization are significant features to burn the injected fuel completely during the combustion stroke in CI (compression ignition) engine. Producing in-cylinder flows such as swirl and tumble by modifying geometry of intake system and combustion chamber has recently received remarkable attention due to enhance combustion efficiency and fuel economy [1]. Swirl and tumble are both organized rotational flows and their rotation axes are parallel to the piston motion and perpendicular to the axis of the cylinder respectively. One of the prominent techniques to produce the organized in-cylinder motion is modifying piston bowl shapes [2-12]. Payri et al. [2] examined the influence of the five piston bowl geometries of a DI diesel engine on the flow field during intake and compression strokes numerically. The results showed that the piston shape had little impact on in-cylinder motion during the intake stroke and the first part of the compression stroke but the piston bowl geometry played a critical role near the top dead center (TDC). Li et al. [3] probed the impact of different piston bowl geometries such as hemispherical combustion chamber (HCC), shallow depth combustion chamber (SCC) and omega combustion chamber (OCC) on combustion characteristics of a biodiesel fueled diesel engine at medium load conditions using the coupled KIVA4–CHEMKIN code. It was concluded that at low engine speed, SCC had greater indicated work, cylinder pressure and heat release rate (HRR) peak in comparison with HCC and OCC whereas at medium and high engine speeds, OCC showed better performance than other bowl geometries because of producing strong squish in a short time. Further, Taghavifar et al. [4] analyzed the influence of modification in piston head shape in terms of bowl movement and the bowl size in four equal increments at constant compression ratio (CR) on the fluid motion, combustion, emission, and engine operation. They found that enhancing outward bowl movement provided better uniformity in air-fuel mixture resulted in higher HRR and peak pressure but combustion initiation delayed to late expansion phase with lower work delivery acting negatively on engine performance and combustion heat production. Calik et al. [8] examined the feasibility of twin swirl initiation in diesel engine using various angular velocities of the initial swirls and injection orientations of fuel sprays to determine the optimum injection settings and swirl ratios to enhance fuel efficiency and reduce emissions. The results showed that there was not so much difference between different swirl ratios at start of injection. Mixing process and combustion showed a similar trend as conventional diesel engines with the increase of horizontal spray angle.

The purpose of this study is to examine effect of twin bowl piston geometry on in-cylinder flow field characteristics and combustion parameters in a direct injection (DI) diesel engines using computational fluid dynamics (CFD) methods during induction, compression and expansion strokes (360°-900°).

2. Computational methods

In this study, numerical simulation of a two valve single cylinder four stroke DI diesel engine was performed using ANSYS Workbench Platform 14. The technical specifications of diesel engine are given in Table 1.

Table 1 The simulation parameters of diesel engine

Type	Two valve single cylinder four stroke DI diesel engine
Bore x Stroke	87.5 mm x 110 mm
Connecting rod length	234 mm
Piston cavity	Hemispherical
Engine speed, CR	1500 rpm, 17.5
Intake valve opening	4.5º bTDC*
Intake valve closing	35.5º aBDC*
Exhaust valve opening	35.5º bBDC
Exhaust valve closing	4.5º aTDC
Number of injector holes	3
Diameter of hole	0.24 mm
Injection pressure	200 bar
Injection time and duration	23º bTDC, 29º
Fuel and cetane number	$C_{14.6}H_{24.8}$, 49
Injected fuel	0.0293 g/cycle
Injected fuel flow rate	0.0091 kg/s

*b: before; a: after.

The geometry with various parts such as the intake and exhaust valves and ports, the cylinder and the piston bowl was constructed and assembled using SOLIDWORKS 2018. Then, the model in parasolid (x_t) format was implied into the ANSYS DesignModeler and the mesh was generated using the proper mesh methods as shown in Fig. 1.

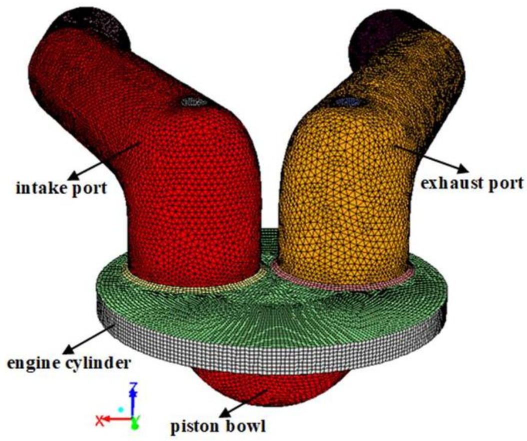

Fig. 1. Different zones in the geometry

The layering approach in ANSYS Fluent was employed for in-cylinder simulation of DI diesel engine. The layering approach is mostly applied to diesel engines with vertical valves. In this method, geometry is decomposed into different parts and they are meshed properly to apply different mesh motion strategies to different regions. In the second step, the simulation parameters related to valves and piston motion, boundary conditions and events to open and close valves are automatically set up by using journal file.

Two configurations were produced to analyze the influence of twin bowl shape on in-cylinder flow and combustion. Twin bowl configurations named TB1 and TB2 are generated by two hemispheres with diameter of 47 mm. TB1 and TB2 are placed along the x-axis and y-axis, respectively, as shown in Fig. 2. CR of TB1 and TB2 configurations and the base model is 17.5.

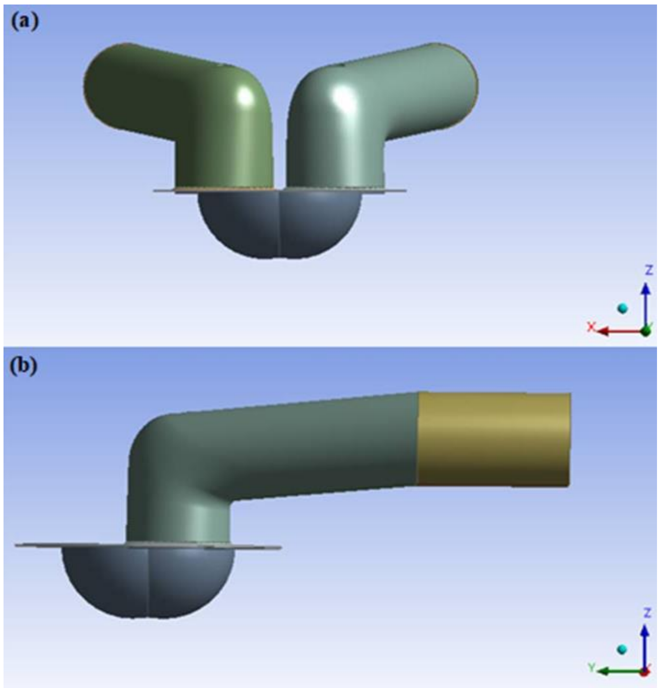

Fig. 2. Twin bowl configurations: a) TB1, b) TB2

2.1 Turbulence model

RNG *k-ε* turbulence model was used for simulating in-cylinder turbulent flow of DI diesel engine. RNG *k-ε* model includes additional term in its dissipation equation. This term enhances the correctness for speedily strained flows. Besides this, the model includes effect of swirl motion on turbulence to improve the accuracy for swirling flows. The turbulent kinetic energy (*k*) and the turbulence dissipation rate (*ε*) are [12]

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left[\alpha_k \mu_{eff}\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho\varepsilon-Y_M+S_k \quad (1)$$

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial x_i}(\rho\varepsilon u_i)=\frac{\partial}{\partial x_j}\left[\alpha_\varepsilon \mu_{eff}\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1e}\frac{\varepsilon}{k}(G_k+C_{3e}G_b)-R_\varepsilon+S_\varepsilon \quad (2)$$

where u_i is the absolute velocity component in direction x_i., ρ is the density and μ_{eff} is the effective viscosity. G_b and G_k denote the generation of turbulence kinetic energy owing to buoyancy and the mean velocity gradients, respectively. Y_M contributes the fluctuating dilatation in compressible turbulence to the overall dissipation rate. R_ε is the term related to the effects of rapid strain and streamline curvature. S_k and S_ε are the source terms. $C_{1\varepsilon}$, $C_{2\varepsilon}$ and C_μ are constants. $C_{1\varepsilon}$=1.42, $C_{2\varepsilon}$=1.68 and C_μ=0.0845, in this study.

2.2 Combustion model.

The species transport model was used to predict in-cylinder combustion process in DI diesel engine. In this model, each species local mass fraction was calculated through the solution of equations describing convection, diffusion and reaction sources. The conservation equation for *i*th species is [13]

$$\frac{\partial}{\partial t}(\rho Y_i)+\nabla.(\rho\vec{v}Y_i)=R_i+S_i-\nabla.\vec{J}_i \quad (3)$$

where Y_i represents the local mass fraction of each species. R_i is the net rate of generation of species because of chemical reaction. In this work, R_i was determined by the eddy-dissipation model originated from early work carried out by Magnussen and Hjertager [14]. S_i is the rate of genaration including the dispersed phase and any user defined source. J_i is the diffusion flux of species for turbulent flows. The ignition delay in the engine was modeled by employing the empirical correlation by Hardenburg and Hase [15]. The Kelvin-Helmholtz (KH) and Rayleigh-Taylor (RT) model was used for the atomization of the injected spray. The KH model was used to describe the initial spray break-up the intact liquid core. RT model was used in combination with the KH model to estimate the secondary break-up of the droplets. In this study, B0 and B1 values are 0.61 and 24, where B0 and B1 are the drop size and the breakup constants, respectively, Point properties such as diameter, position, velocity magnitude, temperature, cone angle and total flow rate were specified for solid-cone injection. The number of droplet collisions and coalescence of droplets were predicted by O'Rourke's model [16].

2.3 Boundary conditions.

The initial boundary conditions of pressure as 1 atm and temperature as 300 K were used for inlet and outlet. The engines walls were described as stationary with no slip condition. The constant temperature boundary condition was set at the walls. Temperature of intake and exhaust ports were 300 K and temperature of piston, cylinder head and cylinder wall were 400 K. The fully developed flow was assumed.

3. Result discussions

In the mesh independent study carried out by the previous work [17], CFD results of in-cylinder pressure obtained by the mesh size of 730734 cells were good agreement with experimental data from Jaichandar et al. [18]. Therefore, the grid size of 731302 cells when piston close to TDC was considered in this study. The percentage error is nearly 1.2% for the peak cylinder pressure as shown in Fig. 3.

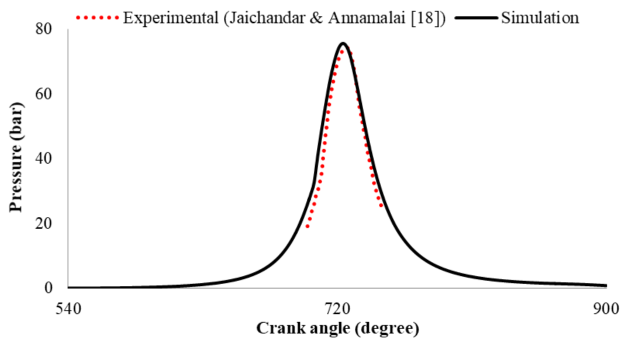

Fig. 3. Comparison of computed and experimental data of in-cylinder pressure

TB1 and TB2 twin bowl configurations were obtained using same mesh methods and cell size. The mesh size of TB1 and TB2 configurations is 730687 and 731140, respectively.

3.1 Influence of twin bowl shape on in-cylinder flow characteristics

The swirl ratio as a dimensionless parameter is used to quantify the swirl flow in the cylinder. The swirl ratio is obtained as [19]

$$SR = \frac{\omega_s}{(2\pi N/60)} \quad (4)$$

where SR is swirl ratio, ω_s is swirl angular speed, rad/s. N is engine speed, rpm. Fig. 4 demonstrates the variation of swirl ratio in the cylinder for twin bowl configurations.

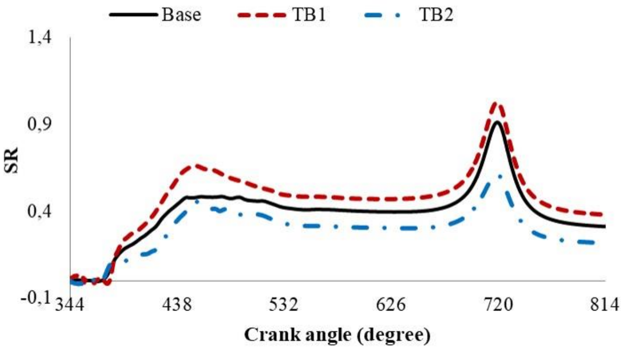

Fig. 4. Variation of *SR* for the twin bowl shape configurations

The trend of the swirl ratio as shown in Fig. 4 was found earlier by Payri et al. [2]. Since the tangential velocity of the swirling motion improved as a result of interaction between the piston cavity and combustion chamber, the maximum swirl ratio was obtained during the late compression stroke. TB1 twin bowl configuration located along to x-axis enhanced the maximum swirl ratio of 1.03 compared to the base model swirl ratio of 0.91 near TDC as shown in Fig. 4.

The tumble ratio as a dimensionless parameter is calculated by

$$TR = \frac{\omega_t}{(2\pi N/60)} \quad (5)$$

where TR is tumble ratio, ω_t is tumble angular speed, rad/s. is engine speed, rpm. Fig. 5 illustrates the change of the tumble ratio about x-axis (*TR-x*) in the cylinder for the twin bowl configurations.

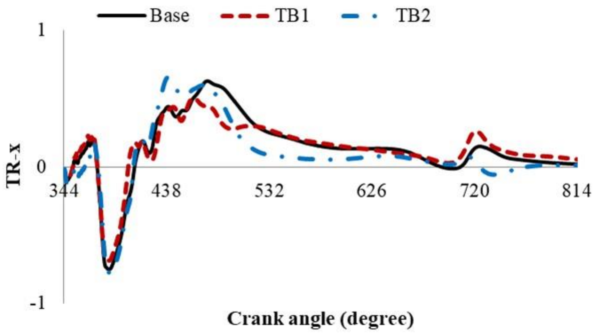

Fig. 5. Variation of *TR- x* for the twin bowl shape configurations

As shown in Fig. 5, TB2 twin bowl configuration located along the y-axis improved *TR-x* of 0.15 near TDC. Fig. 6 shows the change of the tumble ratio about y-axis (*TR-y*) in the cylinder for the twin bowl configurations.

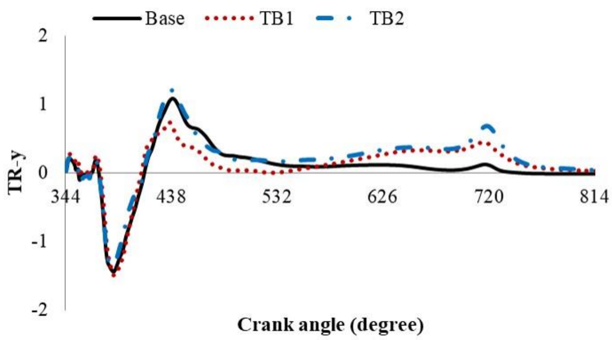

Fig. 6. Variation of *TR-y* for the twin bowl shape configurations

TB1 and TB2 twin bowl configurations developed *TR-y* of 0.45 and 0.68 in comparison to the base model having *TR-y* of 0.12 near TDC, as seen in Fig. 6.
The turbulent kinetic energy, k (m^2/s^2) is defined to be half sum of variances of the turbulent fluctuations

$$k = \frac{1}{2}\left(\overline{u'^2_x} + \overline{u'^2_y} + \overline{u'^2_z} \right) \quad (6)$$

where u'_x, u'_y and u'_z fluctuating velocity in x, y and z flow directions. Fig. 7 illustrates variation of k for the twin bowl configurations.

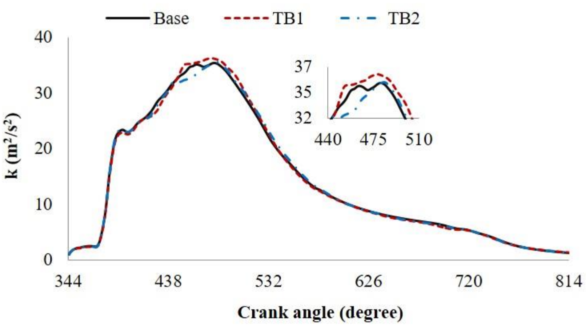

Fig. 7. Variation of k for the twin bowl shape configurations

The maximum turbulent kinetic energy of 36.3 m^2/s^2 is obtained by TB1 twin bowl and it is approximately 2.5% higher compared with the base model as seen in Fig. 7. This result is estimated since TB1 twin bowl configuration significantly improves *SR* during intake stroke in Fig. 4.

3.2 Influences of twin bowl shape on combustion characteristics

The results of in-cylinder pressure for twin bowl configurations are illustrated in Fig. 8.

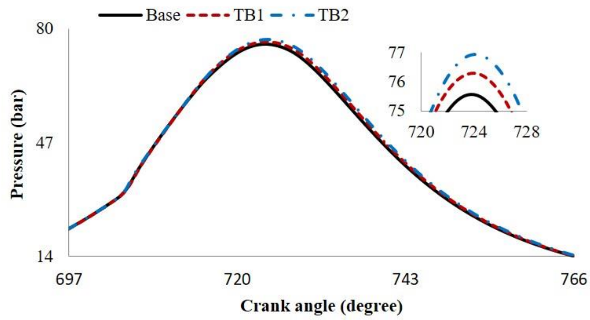

Fig. 8. Variation of pressure for the twin bowl shape configurations

TB1 and TB2 twin bowl configurations enhance the peak pressure of 76.3 and 76.9 bar, respectively, in comparison to the base model peak pressure of 75.6 bar near TDC, as shown in Fig 8. Therefore, TB1 and TB2 configurations elevate the maximum pressure of 0.8 and 1.4%, respectively, compared to the base model. As shown in Fig. 9, TB1 and TB2 twin bowl configurations also increase the maximum in-cylinder temperature of 1685.4 and 1693.6 K, respectively in comparison to the base model with temperature of 1649.3 K.

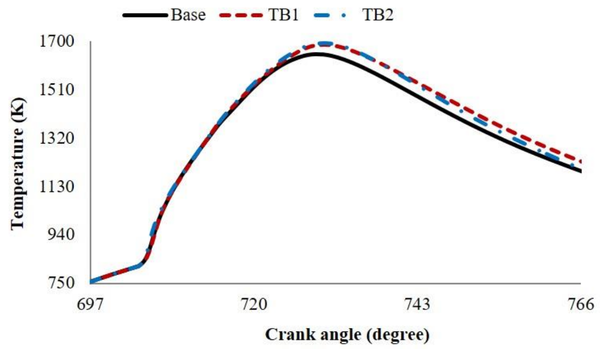

Fig. 9. Variation of temperature for the twin bowl configurations

As seen in Fig. 10, since increasing swirl and tumble ratios close to top of the compression stroke improves the maximum pressure in Fig. 8, TB1 and TB2 configurations enhance heat release rate near TDC on the compression stroke when compared the base model.

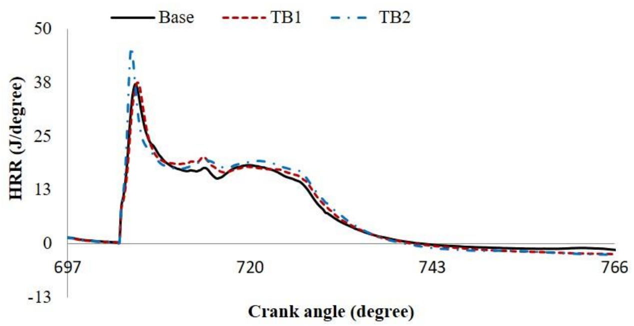

Fig. 10. Variation of HRR for the twin bowl configurations

The indicated fuel conversion efficiency, $\eta_{f,i}$ is ratio of work generated per cycle to the amount of fuel energy supplied [20]. $\eta_{f,i}$ is calculated as

$$\eta_{f,i} = \frac{W_{c,i}}{m_f Q_{LHV}} \tag{7}$$

where $W_{c,i}$ is the indicated work per cycle and it is the summation of the compression stroke work and the expansion stroke work. m_f is the mass of fuel per cycle and Q_{LHV} is the lower heating value of the fuel. $W_{c,i}$ is calculated using MATLAB software in this study. As shown in Fig. 11, TB1 and TB2 configurations enhance $\eta_{f,i}$ of 26.10 and 26.53%, respectively, due to these configurations improving positive work in expansion stroke as compared to the base model $\eta_{f,i}$ of 25.80%.

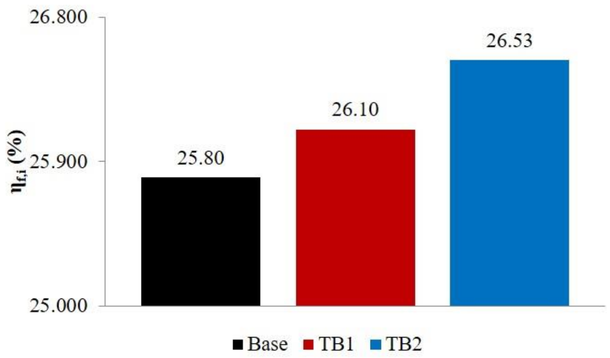

Fig. 11. Variations of $\eta_{f,i}$ for the twin bowl configurations.

Thus, TB1 and TB2 configurations increase $\eta_{f,i}$ of 1.16 and 2.83%, respectively, when compared to the base model.

4. Conclusions

In this study, the influence of two twin bowl configurations on in-cylinder flow and combustion parameters have been investigated numerically during intake, compression and power strokes using SOLIDWORKS, ANSYS Workbench and MATLAB. The following conclusions were highlighted.

1. Variation of swirl ratio follows same trend during simulation for the twin bowl configurations. The maximum *SR* of 1.03 was obtained by TB1 twin bowl configuration.
2. TB1 and TB2 configurations enhanced *TR-y* of 0.45 and 0.68 compared to the base model *TR-y* of 0.12 near TDC.
3. The highest k of 36.3 m^2/s^2 is obtained by TB1 twin bowl configuration and it is nearly 2.5% higher compared with the base model.
4. TB1 and TB2 configurations elevate the maximum pressure by 0.8 and 1.4% and enhance $\eta_{f,i}$ by 1.16 and 2.83%, respectively, owing to enhance positive work in expansion stroke compared to the base model.
5. CFD simulation aids to understand the interaction between in-cylinder fluid motion and combustion by giving detailed information related to intake, compression and combustion processes.

References

[1] Kaplan M. Influence of swirl, tumble and squish flows on combustion characteristics and emissions in internal combustion engine-review, *International Journal of Automotive Engineering and Technologies*, 8, 2, 83-102, 2019.

[2] Payri, F., Benajes, J., Margot, X., and Gil, A. CFD modeling of the in-cylinder flow in direct-injection Diesel engines, *Computers & Fluids*, 33, 995-1021, 2004.

[3] Li J., Yang W.M., An H., Maghbouli A., and Chou S.K. Effects of piston bowl geometry on combustion and emission characteristics of biodiesel fueled diesel engines, *Fuel*, 120, 66-73, 2014.

[4] Taghavifar H., Khalilarya S., and Jafarmadar S. Engine structure modifications effect on the flow behavior, combustion, and performance characteristics of DI diesel engine, *Energy Conversion and Management*, 85, 20-32, 2014.

[5] Prabhakaran P., Ramesh P., Saravanan C.G., Loganathan M. and Gunasekaran E. J. Experimental and numerical investigation of swirl enhancing grooves on the flow and combustion characteristics of a DI diesel engine, *Energy*, 115, 1234-1245, 2016.

[6] Ravi K. and Porpatham E. Effect of piston geometry on performance and emission characteristics of an LPG fuelled lean burn SI engine at full throttle condition, *Applied Thermal Engineering*, 110, 1051-1060, 2017.

[7] Yan B., Tong L., Wang H., Zheng Z., Qin Y. and Yao M. Experimental and numerical investigation of the effects of combustion chamber reentrant level on combustion characteristics and thermal efficiency of stoichiometric operation natural gas engine with EGR, *Applied Thermal Engineering*, 123, 1473-1483, 2017.

[8] Calik A. T.,Taskiran O. O., Mehdiyev R. Numerical investigation of twin swirl application in diesel engine combustion, *Fuel*, 224, 101-110, 2018.

[9] Khana S., Panua R. and Bose P. K. Combined effects of piston bowl geometry and spray pattern on mixing, combustion and emissions of a diesel engine: A numerical approach, *Fuel*, 225, 203–217, 2018.

[10] Leach F., Ismail R., Davy M., Weall A. and Cooper B. The effect of a stepped lip piston design on performance and emissions from a high-speed diesel engine. *Applied Energy*, 215, 679–689, 2018.

[11] Baratta M., Misul D., Viglione L. and Xu J. Combustion chamber design for a high-

performance natural gas engine: CFD modeling and experimental investigation, *Energy Conversion and Management*, 192, 221-231, 2019.

[12] Karthickeyan V. Effect of combustion chamber bowl geometry modification on engine performance, combustion and emission characteristics of biodiesel fuelled diesel engine with its energy and exergy analysis, *Energy*, 176, 830-852, 2019.

[13] ANSYS FLUENT Theory Guide (Release 14.0), November 2011.

[14] Magnussen B. F. and Hjertager B. H. On mathematical models of turbulent combustion with special emphasis on soot formation and combustion. *16th Symposium (International) on combustion, soot formation and growth*, 719-729, the Combustion Institute Pittsburgh, PA, 1976.

[15] Hardenburg H. O. and Hase F. W. An Empirical Formula for Computing the Pressure Rise Delay of a Fuel from its Cetane Number and from the Relevant Parameters of Direct Injection Diesel Engines. *SAE Technical Paper 79049*, 1979.

[16] P.J. O'Rourke. *Collective Drop Effects on Vaporizing Liquid Sprays*. PhD thesis, Princeton University, Princeton, New Jersey, 1981.

[17] Kaplan M., Özbey M. and Özcan H. Numerical Investigation of the Effects of Intake Port Geometry on In-Cylinder Motion and Combustion in Diesel Engine. *The International Journal of Engineering and Science*, 6, 7, 16-26, 2018.

[18] Jaichandar S. and Annamalai K. Effects of open combustion chamber geometries on the performance of pongamia biodiesel in a DI diesel engine. *Fuel*, 98, 272-279, 2012.

[19] Pulkrabek, W.-W. *Engineering Fundamentals of the Internal Combustion Engine*, Prentice-Hall, Upper Saddle River, New Jersey, 1998.

[20] Heywood, J. B. *Internal combustion engine fundamentals*, McGraw-Hill, New York, 1998.

Nomenclature

BDC	Bottom dead center
CFD	Computational fluid dynamics
CR	Compression ratio
DI	Direct injection
HRR	Heat release rate (J/degree)
k	Turbulent kinetic energy (m^2/s^2)
m_f	Mass of fuel per cycle (kg)
N	Engine speed (rpm)
Q_{LHV}	Lower heating value of the fuel (kJ/kg)
SR	Swirl ratio
TDC	Top dead center
TR	Tumble ratio
TR-x	Tumble ratio about x-axis
TR-y	Tumble ratio about y-axis
u'	Fluctuating velocity (m/s)
$W_{c,i}$	Indicated work per cycle (kJ/kg)
α_k, α_ε	Inverse effective Prandtl numbers for k and ε
ε	Turbulence dissipation rate
$\eta_{f,i}$	Indicated fuel conversion efficiency
ω_s	Swirl angular speed (rad/s)
ω_t	Tumble angular speed (rad/s)