EUROPEAN MECHANICAL SCIENCE

e-ISSN: 2587-1110

Research Article

Numerical investigation of different combustion chamber on flow, combustion characteristics and exhaust emissions

İlker Temizer[1], Ömer Cihan[2], Öncel Öncüoğlu[3]

[1]Sivas Cumhuriyet University Technology Faculty, Department of Manufacturing Engineering, 58140, Sivas, Turkey.
[2]Hakkari University, Engineering Faculty, Department of Mechanical Engineering, 30000, Hakkari, Turkey.
[3]Istanbul Technical University, Faculty of Mechanical Engineering, Department of Mechanical Engineering, 34469, Istanbul, Turkey.

ORCID: İ. Temizer (0000-0003-1170-3898), Ö. Cihan (0000-0001-8103-3063), Ö. Öncüoğlu (0000-0002-8377-5375)

Abstract: This study includes numerical analysis of diesel engines with different bowl geometry. Numerical analyzes of the diesel engine with asymmetrical bowl geometry were performed in Ansys Forte software. In the study, four-stroke, air cooled, a single-cylinder and direct injection diesel engine were used. It has been tested where the maximum torque is obtained as the operating condition at 2000 rpm. According to the results obtained from the analyzes, the new combustion chamber system geometry provided a 40.3% reduction in soot emissions while NO emissions increased slightly with the 8-cavity bowl geometry created in the chamber compared to the standard combustion chamber system. Increasing air velocity, and turbulent kinetic energy values in the chamber affected the evaporation levels of the fuels. As a result, the improved mixture formation caused a decrease in incomplete combustion products (CO, HC and soot). The new combustion chamber system geometry according to standard combustion chamber system type, an increase of approximately 4.2% occurred in the calculated squish rates. It has been observed that the increase in the bowl surface area causes the combustion, and thus the temperature to spread over a larger area on the piston.

Keywords: Diesel engine; Bowl geometry; Combustion analysis; Fuel spray distribution.

1. INTRODUCTION

In recent years, engine manufacturers have increased their research to reduce emissions due to awareness of air pollution and strict emission rules. In addition, many studies are carried out on the parameters affecting engine performance and combustion. In internal combustion engines, diesel engine produces better fuel consumption, higher exhaust emission value and higher noise than gasoline engine. Low fuel consumption in diesel engines is used in many areas such as electric power generation, transportation and agricultural machinery [1]. There are many parameters such as fuel specifications, operating conditions and engine constructive design, which are the main factors such as efficiency, combustion characteristics and emissions of a compression ignition engine [2].

A fast and better air-fuel mixture is the most important need for reducing exhaust emissions, improving engine performance and combustion characteristics. For the mixing quality of diesel fuel injected into the combustion chamber with air, the geometry of the combustion chamber, the injection parameters, and the characteristics of the air movements should be improved [3]. Combustion chamber geometry has an important place in the design of diesel engines. For a good geometry, the air-fuel mixture should be improved, and for easier evaporation, there should be more air movement in the cylinder in terms of swirl, squish and turbulence [4]. Singh et al. [5] stated that with different combustion chamber geometries, the desired tumble, swirl, squish and turbulence parameters in the combustion chamber can be improved, thereby lowering exhaust emissions and improving engine performance.

Saito et al. [6] compared the bowl geometries conventional and re-entrant in a single-cylinder, four-stroke, direct injection diesel engine. Engine performance, NOx and soot emissions, and combustion parameters were investigated. As a result, the ignition delay is reduced due to the fact that the re-entrant geometry is hotter than the other, the wall where the fuel hits. Moreover, turbulence increased with the increase of in-cylinder air movements, and combustion improved. Yaliwal et al. [7] reported that the Re-entrant type combustion chamber showed maximum performance under 230 bar injection pressure, four holes and 0.25 mm nozzle opening con-

* Corresponding author.
Email: omercihan@hakkari.edu.tr

European Mechanical Science (2023), 7(1): 7-15
https://doi.org/10.26701/ems.1181494
Received: September 28, 2022 — Accepted: December 8, 2022

ditions. Bapu et al. compared Modified Hemispherical Combustion Chamber (MHCC) and conventional Hemispherical Combustion Chamber (HCC) designs by using ANSYS fluent software in a diesel engine. According to the results obtained, it was seen that the mixture formation was better when the flow movements at different positions of the piston were examined compared to the HCC [8]. Optimum bowl geometry improves air/fuel mixture formation, and reduces rich mixing zones [9]. Both NO_x emissions, and local high temperatures can be controlled by avoiding in-cylinder rich mixing zones [10]. Otherwise, Şener et al. [11] observed which the geometry of the piston bowl has a significant effect on the in-cylinder pressure, in-cylinder temperature, heat release, and exhaust emission values in used engine. In the numerical study using CONVERGE software, DA, DB, DC, DD, DE and DF geometries were used. DF and DE geometries have higher heat release rate, in-cylinder pressure and temperature values due to the main radius of bowl. However, NOx emissions have also increased.

Li et al. [12] numerically and experimentally compared the geometry of the double swirl combustion system (DSCS) and lateral swirl combustion system (LSCS) in a compression ignition engine at different speeds. The influence of LSCS geometry on fuel economy, and soot emissions was investigated. LSCS provided 2.8–4.1g / kWh fuel economy compared to DSCS, and soot exhaust emissions decreased in the range of 69–75%. The bowl shape of the piston directly affects the factors of swirl, squish, tumble, turbulence and flame velocity. Therefore, it causes changes in combustion characteristics and exhaust emissions [13-15].

Air movements in internal combustion engines have a significant effect on fuel/air mixture formation. In this study, a total of eight (8) bowls created on the piston will increase the airflow both in the horizontal and vertical axis. Also, the multi-cavity spreading of the cavity on the piston head will cause the combustion hot surface area to increase. Directing the fuel assembly by striking it against the combustion chamber surface also enables it to evaporate more easily, resulting in reduced ignition delay times. By injecting the fuel to the point where the two cavities meet, the fuel beam will be directed to different cavities and the combustion will be mixed with more air in more than one region. In addition, it is aimed to reduce the thermal stresses on the piston by preventing regional temperature formations by realizing a greater distribution of combustion on the piston surface with this geometry. Starting the evaporation process before the fuel reaches the center of the cavity prevents the fuel from breaking down at high temperatures and reduces the formation of C (soot) emissions to minimum levels. With this geometry, the fuel assembly sprayed by the injector with four nozzles will hit the wall at the junction of the two cavities and will both evaporate with the effect of temperature and be directed towards the cavity center where there is high air movement. In this case, instead of multi-hole injectors and high injection pressures, full combustion can be achieved by using a hole injector with different cavities and low injection pressures which makes it possible to significantly reduce the incomplete combustion products C (PM- particulate matter), CO and HC values.

In this study, a new combustion chamber system (NCCS) with different geometry was analyzed using Ansys Forte software. The obtained results were compared with the standard combustion chamber system (SCCS). The diameter of the piston's bowl bottom base was numerically investigated for R3 =3 mm (NCCS), R3=4 mm (NCCS_V1) and R3 =5 mm (NCCS_V2) in geometry, and some results have been obtained. The effects of this diameter in the bowl geometry on in-cylinder pressure and temperature, instantaneous and cumulative heat release rate, swirl ratio, turbulent kinetic energy and exhaust emissions (CO, CO2, soot, UHC and NO) were investigated. Moreover, the flow properties (temperature, velocity and TKE changes) of the bowl geometries were researched.

2. NUMERICAL STUDY

In order to determine the boundary conditions in a model, it must be tested under different operating conditions. Here, mentioned model was created according to the main dimensions and technical features of the four-stroke, direct injection ANTOR 3 LD 510 diesel engine. The technical data of the engine is given in Table 1. Ansys Forte software was used for the model. In-cylinder heat, flow, mixing, and combustion characteristics were investigated with this software. While NCCS and SCCS geometries were created in the program, the compression ratio of 17.5 (compression ratio of the engine) was kept constant. Generally, air flow movement from the TDC to the BDC decreases in engines. This has a great effect on the formation of the mixture. The NCCS bowl geometry consists of eight pockets. Boundary conditions in Table 2 were used. A detailed reduced chemical combustion mechanism, which has been accepted in many studies in the literature, has been used for make the exhaust emissions closer to reality. RNG k-epsilon turbulence model was used as it is more suitable for combustion analysis. A macro-dimensional examination was made, and it gave results in a short time compared to other models in terms of solution time with this model. This model is the most preferred type in the literature [16, 17]. Adaptive Collision Mesh model was used for droplet collision. One of the biggest advantages of this model is that it eliminates the dependency on the network structure. KH-RT hybrid model was applied for the breakup of the droplets, and the KIVA-based wall collision model was used for the droplets hitting the wall.

The gas force distribution in pockets has become more balanced with NCCS. No changes have been made to the structure and location of the injector, and the original injector data has been added to the software. Injector cone projection was applied according to SCCS type. One of the main criteria in the new design was the eight pocket

 https://doi.org/10.26701/ems.1181494

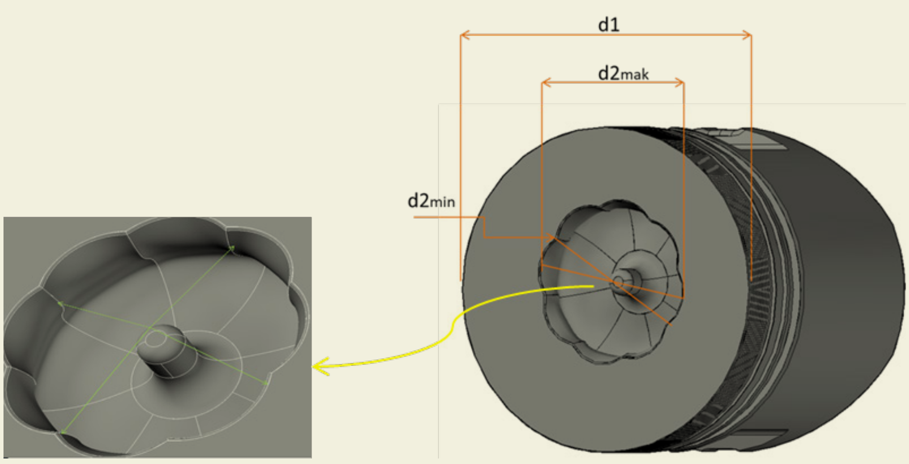

Figure 1. Piston bowl and fuel injection direction

Table 1. Technical properties of the test engine

Engine Name	Antor 3 LD 510
Engine Type	Four stroke, air-cooled, single-cylinder and direct injection diesel engine
Piston displacement	510 cm^3
Stroke x Bore	90 x 85 (mm x mm)
Compression ratio	17.5:1
Power	6.6@3000 (kW)
Torque	32.8@2000 (Nm)
Injection angle	126°
Injector hole number	4

Table 2. Simulation boundary conditions

Number of cylinders	Single cylinder
Type of cooling	Air-cooled
Bore	85 mm
Compression ratio	17.5
Crank radius	42 mm
Number of injection nozzle	4
Injection Timing (start and stop)	705° and 729° CA
Injection spray angle	160°
Injection rate (mass)	5.11e-6 kg
Engine speed	2000 rpm
Air inlet temperature	293.15 (K)
Air inlet pressure	1 (bar)
Fuel injection temperature	330.15 (K)
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	575.15 (K)
Cylinder wall temperature	475.15 (K)

radii placed from the top of the piston. It is aimed to plaster the fuel on the wall, and to evaporate faster by making use of the wall temperature, and higher swirl with the NCCS geometry. It was observed that the mixture formation in the NCCS geometry was affected by the swirl rate at the end of the intake stroke.

Piston design in reciprocating engines is one of primary importance subject areas among engine parts. Many parameters such as compatibility with engine cylinders, combustion effect, and flow direction feature can be studied. In the present paper, it is aimed to find out the effect of changes to be made on the NCCS main geometry. In the new piston geometry designed in this direction, the geometric parameters of SCCS such as piston outer diameter, skirt part, and bowl depth were kept constant. In Fig. 1, some geometric lengths of NCCS are given. The bowl diameter, which is called d2 in SCCS geometry, has two different values in NCCS as maximum and minimum. Gradual narrowing of the bowl diameter of the NCCS is observed. It is known that small throat diameter, and swirl ratios affect NO emissions and heat transfer rate [18]. In order to create stronger flow mobility, a design that will increase the gradual flow rate in the throat design is presented. The zones where the fuel particles hit the wall, and their distribution are visually given in numerical analysis. The injection axis of the fuel particles in the bowl, and the zones of impact on the wall are shown in Fig. 1. It causes less energy loss with fuel sprayed on a wall with a round lip compared to a flat geometry [19]. The diameter of the piston's bowl bottom base was investigated, and the effect of the change in this diameter was examined.

3. RESULTS and DISCUSSION

Three different bowl bottom radius values were determined for NCCS in the Ansys Forte program, and the most ideal geometry was selected from the results, and compared with the SCCS values. Especially in hollow geometries, the bowl bottom radius has a great importance in both air and fuel distribution. For this reason, the effect of different radius values on the geometry consisting of eight pockets was investigated. The diameter of the piston's bowl bottom base was numerically investigated for R_3 =3 mm (NCCS), R_3=4 mm (NCCS_V1) and R_3 =5 mm (NCCS_V2) in geometry, and some results have been obtained. It has caused a decrease in the swirl rate around TDC for all combustion chamber models, especially due to its cavity structure. It can be said that the swirl ratio of the standard geometry is achieved, and combustion and emissions are reduced significantly with the reduction of this radius (for R3 =3 mm). Numerical combustion analysis results of NCCS and SCCS are given in Fig. 2, Fig. 3, Fig. 4 and Fig. 5.

In all analyzes with NCCS geometry, it can be said that

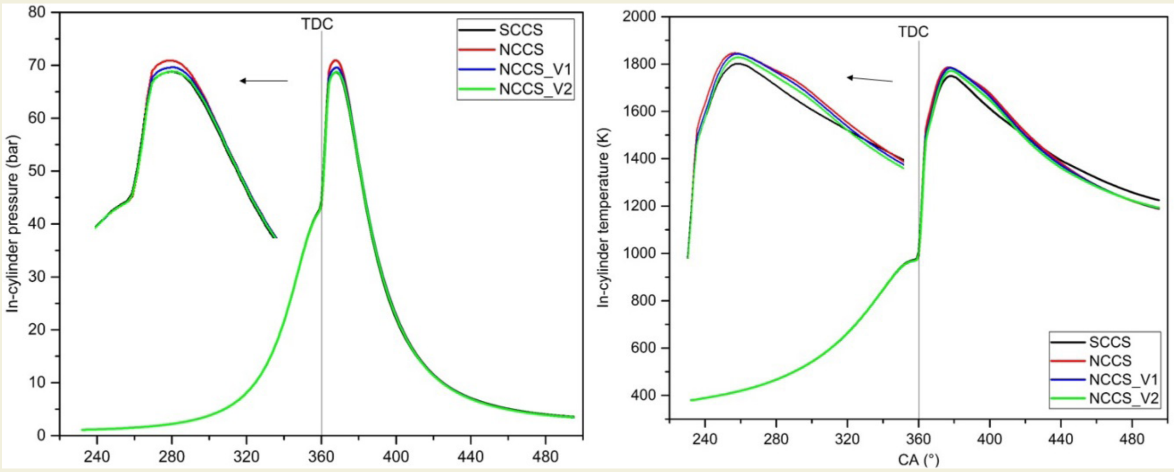

Figure 2. Change of in-cylinder pressure and temperatures with crank angle for NCCS and SCCS geometries

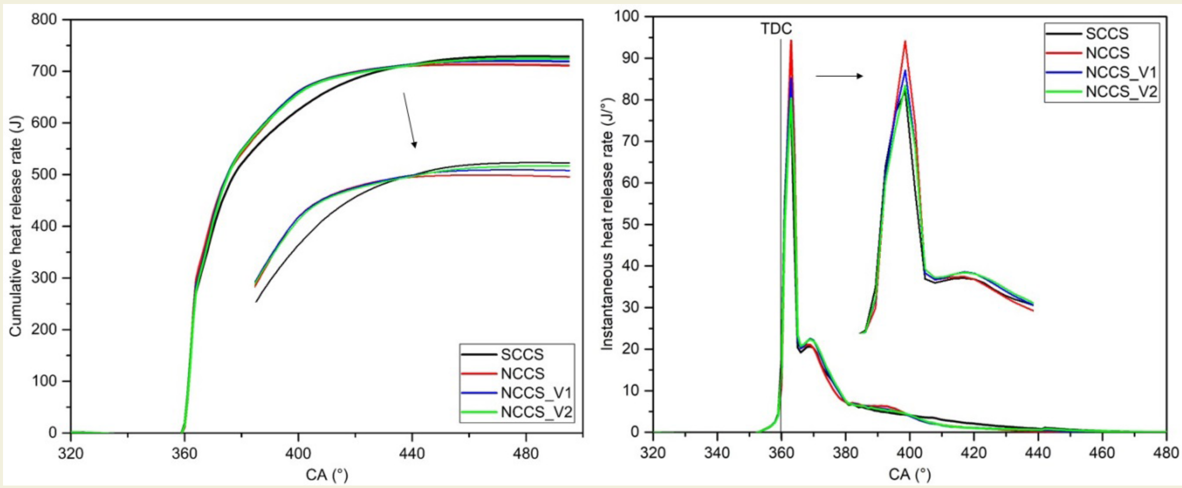

Figure 3. Change of cumulative and instantaneous heat release rates for NCCS and SCCS geometries

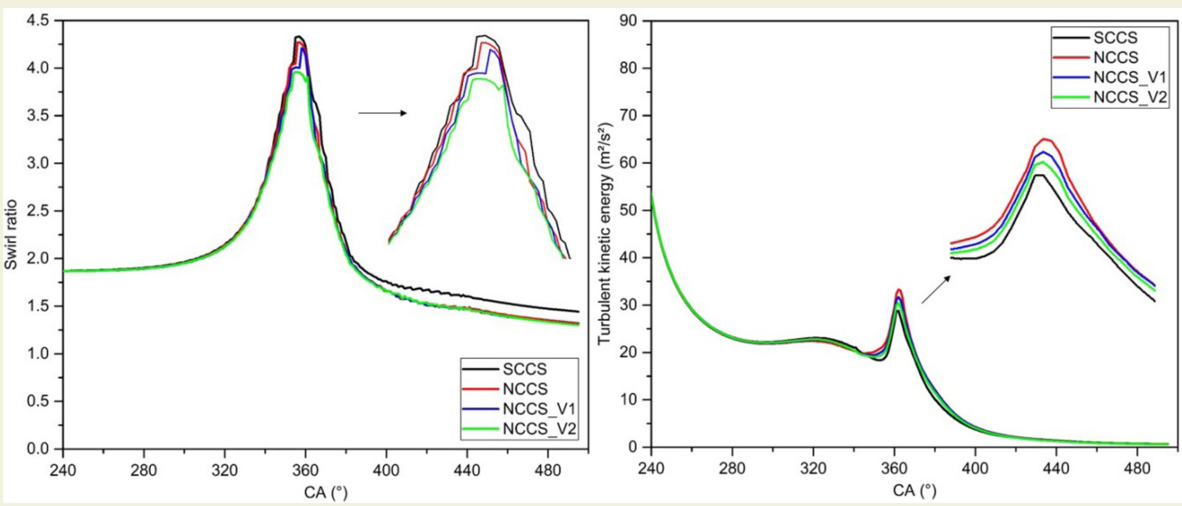

Figure 4. Change of turbulent kinetic energy and swirl ratio with crank angle for NCCS and SCCS geometries

https://doi.org/10.26701/ems.1181494

the maximum cylinder temperature and pressures are slightly higher than SCCS (Fig. 2). In these combustion chambers, the increase in in-cylinder temperatures in parallel with the improvement of mixture formation is thought to be the most importance factor in the increase of NO emissions. In NCCS geometries, it can be said that improvement is achieved in other incomplete combustion products (CO, UHC and CO2) other than NO. Plastering the fuel sprayed on the combustion chamber wall may cause the droplets to evaporate in a shorter time by taking advantage of the wall temperature. In the analyzes made for different radius values of the inner wall, it was determined that in particular NO emission decreased depending on the change in the distance in the penetration depth of the fuel. When the exhaust emission (soot, CO and HC) formations in these models were examined, the NCCS compared to the SCCS was determined that there were significant reductions (Fig. 5). The NCCS compared to the SCCS, it is seen that the most reduction in in-cylinder soot (Particular matter) formation occur-

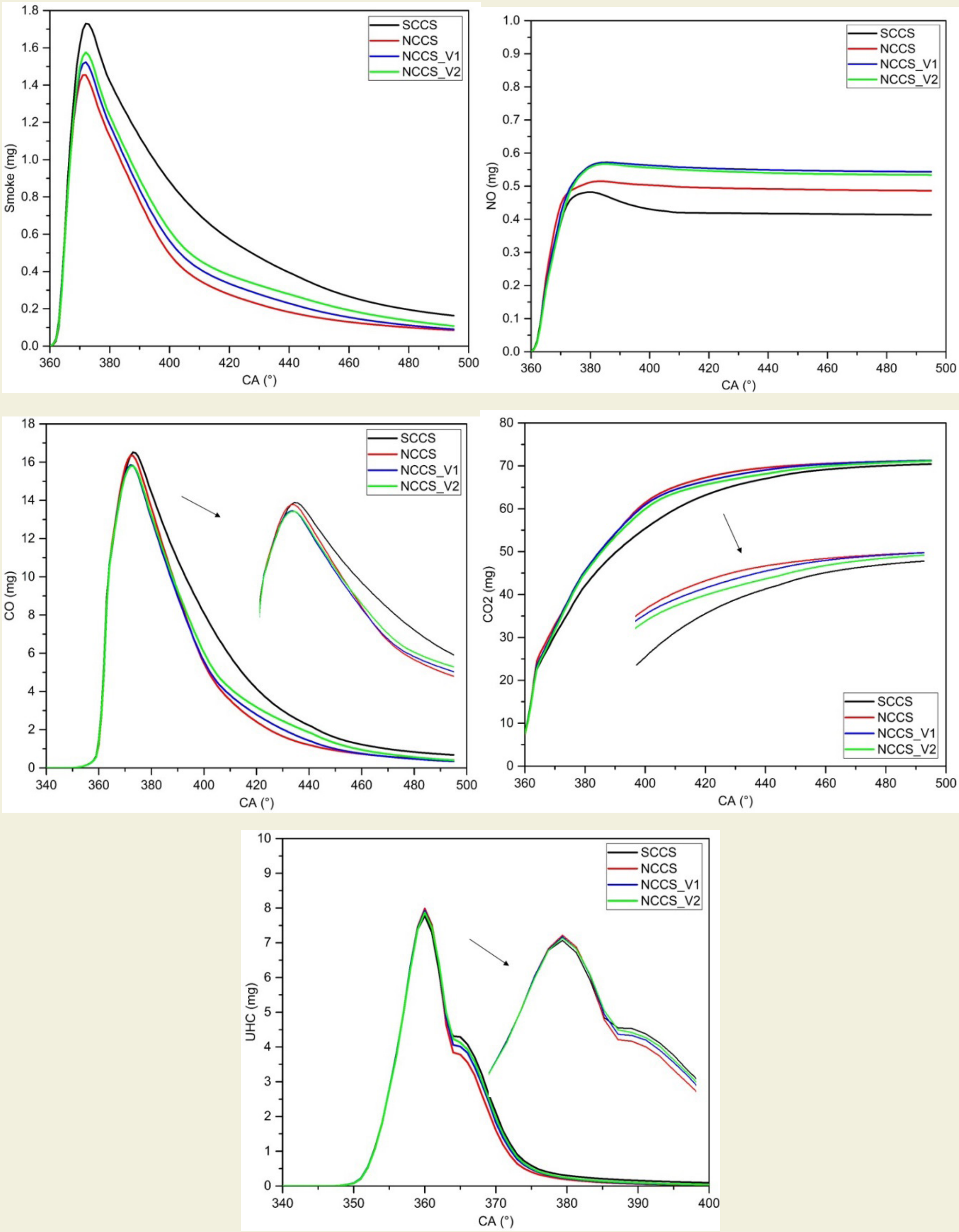

Figure 5. Change of exhaust emission values (Smoke, NO, CO, CO2 and UHC) obtained for NCCS and SCCS geometries

red. According to CFD analyses, the NCCS compared to the SCCS when the amount of soot generated in a cycle at the moment the exhaust valve is opened, a reduction of 40.3% was achieved. Improvement of mixture formation, and wall-fuel interaction is thought to be important in this reduction. As a matter of fact, CO_2 emissions increased in parallel with the improvement of combustion (Fig. 5).

Fig. 6 and Fig. 7 show some analyzes inside the combustion chambers of two different pistons at 720° CA (upper) and 730° (lower) CA. When the temperature/fuel spray distributions are examined, it is seen that the swirl influence in-chamber is important (Fig. 6). Swirl affects the distribution of the fuel droplets in the chamber, the combustion efficiency, and therefore exhaust emissions are affected. Fuel droplets seen in numerical analyzes represent liquid fuel. SCCS seems to be more intense regionally than NCCS (located on the left side of the figure). When the liquid particle distribution of NCCS, which

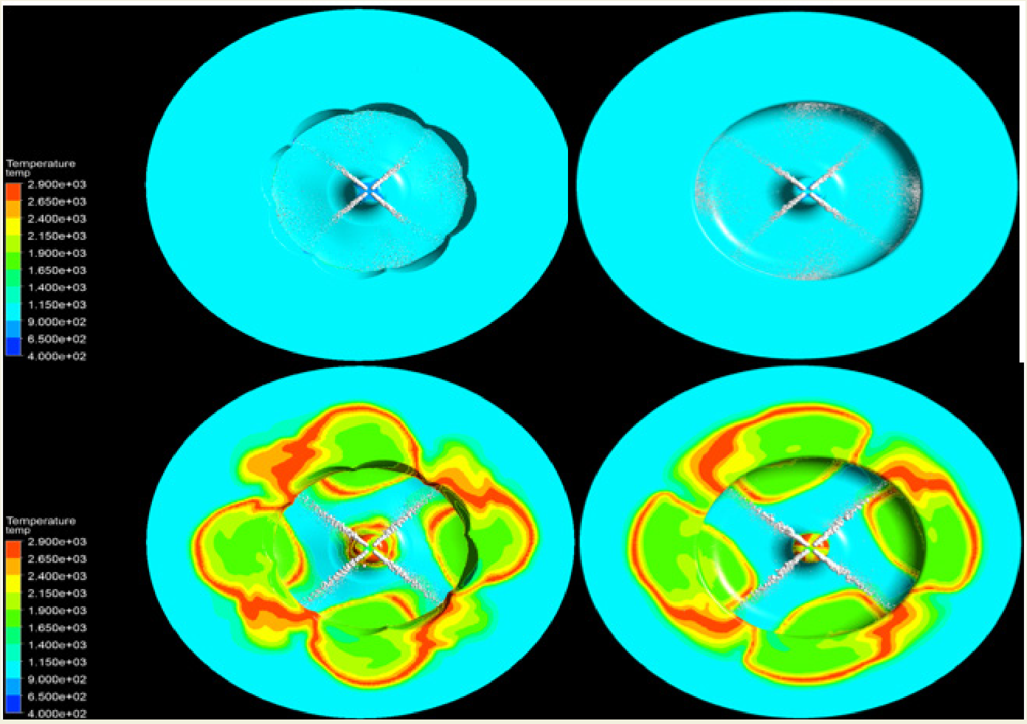

Figure 6. Temperature changes of NCCS and SCCS geometries

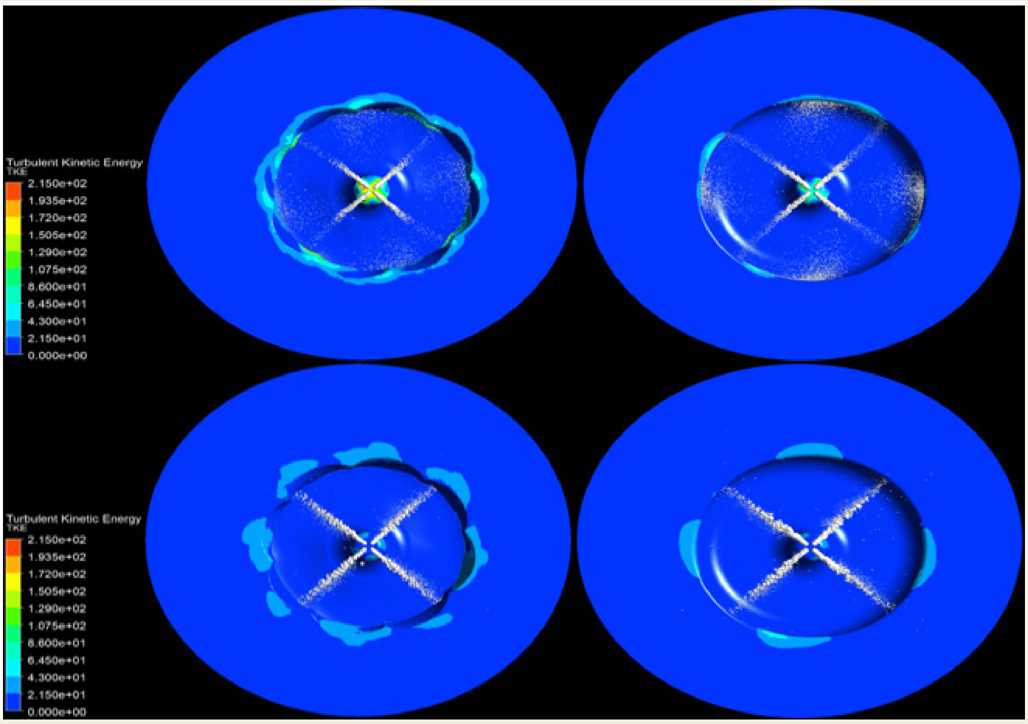

Figure 7. TKE changes of NCCS and SCCS geometries

has eight (8) cavity structures, is examined, a less, and more homogeneous distribution is observed compared to SCCS. This situation is thought to be a result of increasing the evaporation level by taking advantage of the wall temperature of the fuel. An improvement can be seen in the mixture as the amount of steam in the vapor + liquid fuel increases. In addition, some differences in combustion may occur. This is also of great importance in terms of emissions. The energy loss caused by the liquid fuel hitting a flat wall affects the fuel and flame distribution. When the TKE distribution images for different combustion chambers are examined, it can be said that the flow rate is more realized for NCCS geometries (Fig. 7). Although the swirl ratios in the swirl graphs are similar for both combustion chambers, it can be mentioned that the squish mobility is reflected in the TKE (Fig. 7). As a matter of fact, this issue is reflected in the velocity distribution graphs for both crank angles.

Swirl and tumble are called the movements of the air or air/fuel mixture in the chamber. While the swirl movement develops parallel to the cylinder axis, the tumble takes place perpendicular to the movement axis of the piston. Squish movements also produce a tumble flow when the piston reaches the TDC. The dimensionless parameter known as SQ is used to measure squish mobility.

For the squish rate;

$$SQ=A_{squish}/A_{total}$$

In other words, it is the ratio of the top surface area of the piston crown to the cylinder surface area of the total piston. Squish level is determined by the gap between the piston and the head. According to Taylor, the diameter of the gap having a value of less than 0.005 is very important in terms of improvement [20]. The upper surface view of the piston for two different combustion chambers is presented in Fig. 9.

The NCCS compared to the SCCS, an increase in SQ (squish ratio) has been achieved. It can be said that this increase is also reflected in the TKE and velocity distributions obtained in the numerical analysis. Excessive SQ ratio is known to increase knocking tendencies in engines [21]. The absence of such a finding based on the numeri-

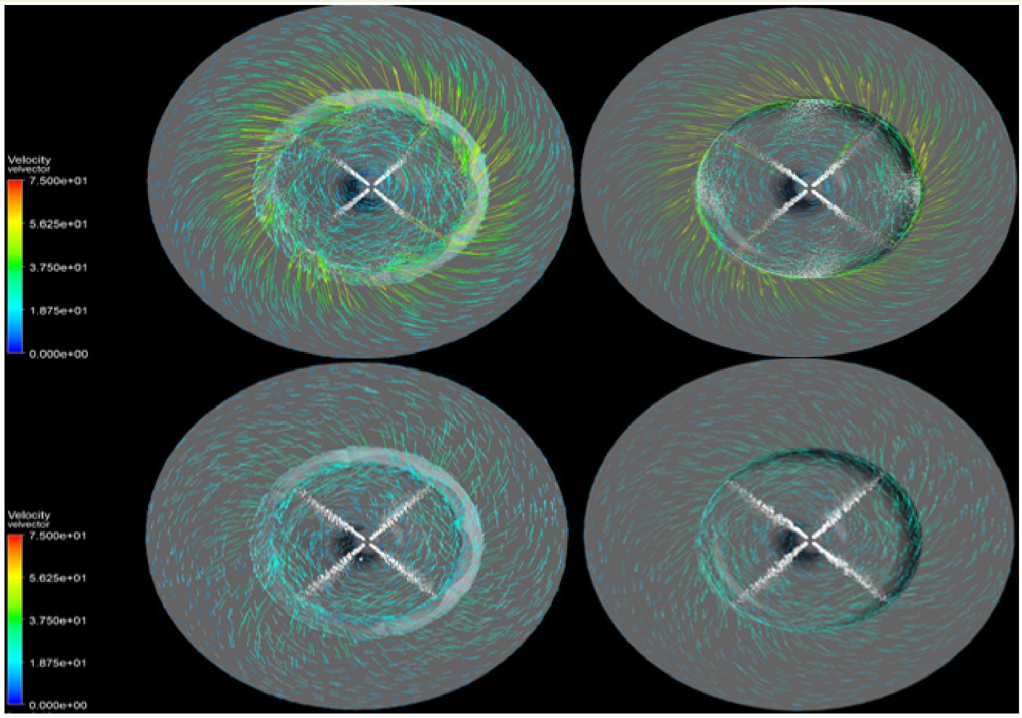

Figure 8. Velocity distributions of NCCS and SCCS geometries

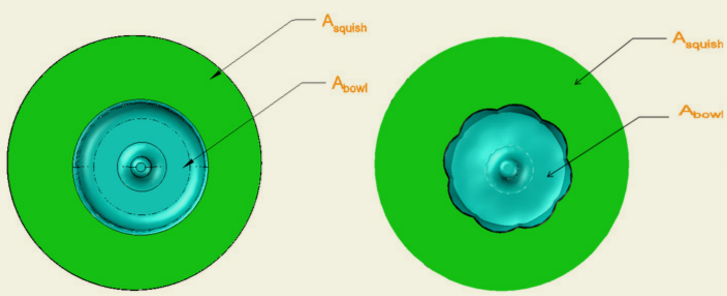

Figure 9. Representation of the upper surface areas of NCCS and SCCS pistons

Table 3. Changes in the surface areas of both pistons

CC Type	A_{squish} (mm²)	$A_{bowl,top}$ (mm²)	A_{total} (mm²)	$A_{bowl,surface}$ (mm²)	SQ
SCCS	4055.6	1618.9	5674.5	3819	0.71
NCCS	4241	1433.5	5674.5	4020	0.74

cal study data shows that the design is acceptable within the limits of conformity. Another parameter accepted in combustion chamber designs is the "k factor" value. This value is known as the ratio of the bowl volume to the total volume when the piston is in TDC. In order to reveal the effect of geometry in the combustion chamber design, no changes were made in compression ratios and k-factor. Table 3 shows the variation of the bowl surface area in the chamber for two different chamber geometries. Increasing the bowl surface area means that the thermal forces on the piston are reduced, and the amount of heat transfer from the wall increases. It will also mean some improvement in the mixture formation of fuel droplets guided by evaporation through the wall. In many studies, it is emphasized that squish movements around the TDC are effective on turbulence [22-25]. More TKE is occurring as seen in the NCCS. This situation occurred as a result of the effect of squish movements. Looking at the velocity analysis, it can be said that the high flow velocity seen in NCCS developed as a result of the combination of squish and swirl flows. Increasing the flame area increases the heat transfer level of the unburned gas [26]. Due to the increased heat transfer, the increase in the temperature of the unburned gas causes the combustion to take place faster, and more fuel to participate in the combustion event. Increases in swirl ratios in engines can result in increased NO emissions [27]. Therefore, another important parameter in piston design was the in-cylinder swirl change. Considering the effect of the swirl ratio on other emissions, a slightly lower NCCS geometry was preferred compared to the SCCS geometry.

4. CONCLUSIONS

The findings obtained in this study, in which heat, flow and combustion parameters are examined in combustion chambers with different geometric designs, are listed in the following items;

- The formation of different base radii and the change of penetration distance changed the evaporation amounts by taking advantage of the wall temperature of the fuel.
- Maximum pressure values showed parallelism with the increase in TKE and flow rate in the cylinder.
- The NCCS according to the SCCS, although the swirl rates of the new combustion chamber are similar, the increases in squish rates are effective parameters in increasing the speed and TKE.
- - The air movement created by eight (8) cavities in the combustion chamber, and its effect on fuel distribution caused a decrease in exhaust emissions (CO, C, HC), and increased NO emissions.
- The NCCS compared to the standard combustion chamber, the increased surface area of the geometric piston containing eight (8) cavities prevented local temperatures, and caused it to spread over a wider area.
- Increasing the bowl surface area on the piston means that the thermal forces are reduced, and the amount of heat transfer from engine increases. This case has caused its formation to create of new research on piston rings and engine wear.

ACKNOWLEDGMENT

This study was supported by the Scientific and Technological Research Council of Turkey (TUBITAK) within the scope of project 120M143.

NOMENCLATURE

BDC	Bottom dead center
C	Carbon
CA	Crank angle
CC	Combustion chamber
CFD	Computational fluid dynamics
CO	Carbon monoxide
CO2	Carbon dioxide
NCSS	New combustion chamber system geometry
NO	Nitrogen oxide
SCCS	Standard combustion chamber system
SQ	Squish ratio
TDC	Top dead center
TKE	Turbulent kinetic energy
UHC	Unburned hydrocarbon

REFERENCES

[1] Jaichandar, J., Annamalai, K. (2012). Influences of re-entrant combustion chamber geometry on the performance of Pongamia biodiesel in a DI diesel engine. Energy, 44(1): 633-640. https://doi.org/10.1016/j.energy.2012.05.029

[2] Challen, B., Barnescu, R. (1999). Diesel engine reference book. England: Society of Automotive Engineers; Bath Press.

[3] Montajir, R., Tsunemoto, H., Ishitani, H., Minami, T., (2000). Fuel spray Behavior in a small DI Diesel engine: effect of combustion chamber geometry. SAE Technical Paper, 2000-01-0946: 1-12. https://doi.org/10.4271/2000-01-0946.

[4] Kalay, İ. (2011). Investigation of piston bowl in diesel engines, M.Sc thesis, Istanbul Technical University, Institute of science and technology, Istanbul, Turkey.

[5] Varun, Singh, P., Tiwari, S.K., Singh, R., Kumar, N., (2017). Modification in combustion chamber geometry of CI engines for suitability of biodiesel: A review. Renewable and Sustainable Energy Reviews, 79: 1016-1033. https://doi.org/10.1016/j.rser.2017.05.116

[6] Saito, T., Daisho, Y., Uchida, N., Ikeya, N. (1986). Effects of Combustion Chamber Geometry on Diesel Combustion. SAE Technical Paper, 861186: 71-81. https://doi.org/10.4271/861186.

[7] Yaliwal, V.S., Banapurmath, N.R., Gireesh, N.M., Hosmath, R.S., Donateo, T., Tewari, P.G. (2016). Effect of nozzle and combustion chamber geometry on the performance of a diesel engine operated on dual fuel mode using renewable fuels. Renewable Energy, 93: 483-501. https://doi.org/10.1016/j.renene.2016.03.020

[8] Bapu, B.R.R., Saravanakumar, L., Prasad, B.D. (2017). Effects of combustion chamber geometry on combustion characteristics of a DI diesel engine fueled with calophyllum inophyllum methyl ester. Journal of the Energy Institute, 90(1): 82-100, https://doi.org/10.1016/j.joei.2015.10.004

[9] Sener, R., Yangaz, M.U., Gul, M.Z. (2020). Effects of injection strategy and combustion chamber modification on a single-cylinder diesel engine. Fuel, 266: 1-15. https://doi.org/10.1016/j.fuel.2020.117122

[10] Dimitriou, P., Wang, W., Peng, Z. (2015). A piston geometry and n ozzle spray angle investigation in a DI diesel engine by quantifying the air-fuel mixture. International Journal of Spray and Combustion Dynamics, 7(1): 1-24. https://doi.org/10.1260/1756-8277.7.1.1

[11] Sener, R., Ozdemir, M.R., Yangaz, M.U. (2019). Influence of piston bowl geometry on combustion and emission characteristics. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, 233(5): 576-587. https://doi.org/10.1177/0957650919854637

[12] Lia, X., Chena, Y., Sub, L., Liua, F. (2018). Effects of lateral swirl combustion chamber geometries on the combustion and emission characteristics of DI diesel engines and a matching method for the combustion chamber geometry. Fuel. 224: 644-660. https://doi.org/10.1016/j.fuel.2018.03.063

[13] Shivashimpi, M.M., Alur, S.A., Topannavar, S.N., Dodamani, B.M. (2018). Combined effect of combustion chamber shapes and nozzle geometry on the performance and emission characteristics of C.I. engine operated on Pongamia. Energy, 154: 17-26. https://doi.org/10.1016/j.energy.2018.04.097

[14] Zhang, L., Ueda, T., Takatsuki, T., Yokota, K., (1995). A study of the effects of chamber geometries on flame behavior in a DI Diesel engine. SAE Technical Paper, 952515: 1-9. https://doi.org/10.4271/952515.

[15] Mittal, G., Gwalwanshi, M. (2021). Influence of reentrant piston bowl geometry on combustion in CI engine. Materials Today: Proceedings, 46(20): 11011-11014. https://doi.org/10.1016/j.matpr.2021.02.098

[16] Liu, D., Li, X., Shang, H., Xie, L., Chen, Y., Chang, J. (2021). Combustion performance and fuel injection timing adaptability of a lateral swirl combustion system for direct injection diesel engines. Fuel, 286: 1-15. https://doi.org/10.1016/j.fuel.2021.120663

[17] Temizer, İ., Cihan, Ö., Eskici, B. (2020). Numerical and experimental investigation of the effect of biodiesel/diesel fuel on combustion characteristics in CI engine. Fuel, 270: 1-9. https://doi.org/10.1016/j.fuel.2020.117523

[18] Amate A.P., Khairnar, H.P. (2015). Disquisition on Diesel Engine Emissions and Piston Bowl Parameters. International Advanced Research Journal in Science, Engineering and Technology, 2(7): 42-49. doi: 10.17148/IARJSET.2015.2710

[19] Sreedharan, S.N., Krishnan, R. (2018). Development of tool to design piston bowl considering spray parameters to reduce emissions. Materials Science and Engineering, 396: 1-8: doi:10.1088/1757-899X/396/1/012055.

[20] Kaplan, M. (2019). Influence of swirl, tumble and squish flows on combustion characteristics and emissions in internal combustion engine-review. International Journal of Automotive Engineering and Technologies (IJAET), 8(2): 83-102. https://doi.org/10.18245/ijaet.558258

[21] Towers, J., Hoekstra, R. (1998). Engine knock, A renewed concern in motorsports - A literature review. SAE Technical Paper, 983026: 1-17. https://doi.org/10.4271/983026.

[22] Payri, F., Benajes, J., Margot, X., Gil, A. (2004). CFD modeling of the in-cylinder flow in direct-injection Diesel engines. Computers & Fluids, 33: 995-1021. https://doi.org/10.1016/j.compfluid.2003.09.003

[23] Gunabalan, A., Ramprabhu, R. (2009). Effect of piston bowl geometry on flow, combustion and emission in DI engines-a CFD approach. International Journal of Applied Engineering Research, 4(11): 2181-2188.

[24] Raj, A.R.G.S., Mallikarjuna, J.M., Ganesan, V. (2013). Energy efficient piston configuration for effective air motion – A CFD study. Applied Energy, 102: 347-354. https://doi.org/10.1016/j.apenergy.2012.07.022

[25] Harshavardhan, B., Mallikarjuna, J.M. (2015). Effect of piston shape on in-cylinder flows and air-fuel interaction in a direct injection spark ignition engine - A CFD analysis, Energy, 81: 361-372. https://doi.org/10.1016/j.energy.2014.12.049

[26] He, Y. (2007). Effect of intake primary runner blockages on combustion characteristics and emissions in spark ignition engines, PhD Thesis, University of Ohio, USA.

[27] Genzale, C., Wickman, D., Reitz, R.D. (2006). An advanced optimization methodology for understanding the effects of piston bowl design in late injection low-temperature Diesel combustion, In: Proceedings of THIESEL 2006 conference on "thermo and fluid dynamic processes in diesel engines", Valencia-Spain.