INTERNATIONAL JOURNAL OF AUTOMOTIVE SCIENCE AND TECHNOLOGY

2021, VOL. 5, NO: 3, 206-213 www.ijastech.org e-ISSN: 2587-0963

Heat and Flow Analysis of Different Piston Bowl Geometries in a Diesel Engine

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, heat and flow in a four cylinder direct injection diesel engine combustion chamber has been simulated by using AVL Fire ESE Diesel. In order to clarify the effect of the combustion chamber geometry on the flow area properties, the bowl geometric shape of two different pistons, one of the combustion chamber as a standard, was taken into account in the flow analysis. Standard combustion chamber and modified combustion chamber geometries have been compared. The simulation results showed that the bowl shapes of the combustion chambers are quite effective on temperature-spray droplet distribution, turbulence kinetic energy distributions, turbulence velocity distributions and laminar flame speed distributions at the end of the compression stroke. The fuel reaches the cylinder wall more easily and then, the temperature distribution in the chamber is lower as a result of evaporation of the fuel with modified combustion chamber. Average turbulent kinetic energy value in the MCC type combustion chamber is 10.53 m^2/s^2, in the standard combustion chamber type combustion chamber this value is 8.35 m^2/s^2 at 720° CA. Turbulence velocity distribution is spread over a wider area in the modified combustion chamber geometry. As a result of the large area of turbulence, the laminar flame velocity has also increased in this chamber geometry.

Keywords: In-cylinder flows, Bowl geometry, Diesel engine, AVL Fire

Research Article

https://doi.org/10.30939/ijastech..929299

Received 28.04.2021
Revised 22.05.2021
Accepted 07.06.2021

* Corresponding author
Ömer Cihan
omercihan@hakkari.edu.tr
Address: Hakkari University, Mechanical Engineering Department, Hakkari, 30000, Turkey.
Tel:+903122028653

1. Introduction

In recent years, as a result of engine performance and strict emission standards, researchers have intensified studies to improve the flow and air movement in the combustion chamber [1]. During fuel injection, air movements, air-fuel mixture, spray dynamics, combustion and emission formation are especially important in the top dead center (TDC) region. Moreover, to air movements, engine speed, combustion chamber shape, spray dynamics and alternative fuel use also affect combustion in diesel engines [2, 3]. It is very difficult and expensive to obtain detailed information about mixture formation and combustion processes and to examine the parameters affecting these processes by experiments. Therefore, multidimensional modeling approach has an important place in engine design [4].

In internal combustion engines, the combustion chamber fluid dynamics is critical in the combustion process [5, 6]. Buyukkaya [7] analyzed the turbulent flow field in the combustion chamber of a direct injection (DI) diesel engine using computational fluid dynamics (CFD). The effect of three different combustion chamber geometries on flow field properties has been investigated. As a result, it has been observed that the combustion chamber bowl geometries greatly affect the pressure, velocity and temperature distributions at end of the compression. Kaplan [8] has emphasized that swirl, tumble and squish flows generally improve in-cylinder turbulence. Stephenson et al. [9] researched the phenomenon of swirl in a direct injection diesel engine. KIVA software was used in the study. It has been observed that very big swirl rates have a considerable effect on the ignition delay. Gunabalan et al. [10] focused on the effect of three different bowl geometries on chamber flow in a direct injection diesel engine using STAR CD software. As the piston moves towards the TDC, the bowl geometry has a significant influence on the air flow, and at the same time, better mixing, better combustion and lower NO_x emission has been achieved thanks to the swirl action. Zheng et al. [11] analyzed the performance of the gasoline direct injection (GDI) engine using AVL-FIRE software. The effects of four different piston geometries on mixture formation and combustion were compared. A piston having a smoother top surface improved turbulence kinetic energy during ignition, thus accelerating combustion and increasing in-cylinder temperature and pressure owing to tumble motion. According to Chu et al. [12], higher tumble motion increases latter combustion rate and faster overall burn time is achieved in diesel engine. Moreover, thermal efficiency can be improved at high loads. Squish, a radial movement towards the centerline of the cylinder, it lead to tumble motion as the piston approaches TDC, and then creates turbulence by breaking small-scale eddies similar to swirl and tumble motion [8, 13]. Perini et al. [14] examined the in-cylinder flow structure with three different piston geometries in a diesel engine. These three piston geometries, first and second featuring a conventional re-entrant bowl, either with or without valve cut-outs and the third featuring a stepped-lip bowl. FRESCO code was used for modeling. The results showed that conventional re-entrant bowls have stronger flow separation at intake stroke, prevent bowl swirl, more axisymmetric squish mechanism and less tilted swirl than other piston geometries. In the present work the combine effects of spray angle and the piston bowl geometry on mixing, combustion and emission characteristics of a direct injection diesel engine have been analyzed numerically. Khan et al. [15] evaluated the effects of spray angles (150°, 155°, 160° and 165°) and piston bowl geometries (Toroidal Re-entrant Combustion Chamber (TRCC), Toroidal Combustion Chamber (TCC) and Hemispherical Combustion Chamber (HCC)) on mixing, combustion and emission characteristics in a diesel engine with a diffraction jet using AVL FIRE software. According to the analysis results, it was understood that the spray angle significantly affected the mixing and combustion process for all three bowl geometries and in the engine with TRCC type combustion chamber gave better performance. Sener et al. [16] investigated the effects of six different piston bowl geometries (Bowl geometries called DA, DB, DC, DD, DE and DF) on combustion and emission characteristics using the CFD code called ANSYS FORTE in a diesel engine. As a result, it has been revealed that the resulting squish and sudden turbulence have a significant effect on in-cylinder pressures, temperatures and heat release rates. In the in-cylinder flow temperature distribution, the average temperature is higher, especially for DE and DF designs. Pressure and velocity distributions also support this result, and there is a more homogeneous distribution.

Li et al. [17] conducted a numerical study on the effect of piston bowl geometries (Hemispherical Combustion Chamber (HCC), Shallow depth Combustion Chamber (SCC), and Omega Combustion Chamber (OCC)) on the combustion and emission characteristics of a diesel engine under medium load conditions. In the study using the KIVA-4 code, the narrow entrance of the combustion chamber can create a powerful squish, especially at high speed, and therefore a better air and fuel mixture is provided. Mohan et al. [18] focused on the effect of shaping the injection rate on the combustion and emissions of a direct injection diesel engine. KIVA-4 code was used in numerical analysis. As a result, higher preload pressure profiles were obtained, moreover better NOx-soot trade-off with injection rate shapes. Aksoy and Yilmaz [19] tested a single-cylinder diesel engine at different engine loads (3.75, 7.5, 11.25 and 15 Nm) and at 2200 rpm, which is the maximum torque speed of the engine. It has been observed that the temperature inside the cylinder at the end of the combustion increased. Lešnik et al. [20] indicated that the effects of mineral diesel fuel and synthetic gas-liquid fuel (GTL) on the injection process, fuel flow conditions and cavitation formation in a modern common-rail injector are examined. A slightly higher average mass flow rate and velocities were obtained for diesel fuel compared to GTL fuel. Bishop et al. [21] discussed emissions and motions within the combustion chamber by using the AVL Fire ESED model. It has been reported that the spray rate shape improves combustion and contributes significantly to the flow in the chamber. Hasan et al. [22] observed the effects of burning velocity and air to liquid mass flow rates. It was observed that the burning velocity decreased. Moreover, air to liquid mass flow rates increased as the burning velocity increased.

In this study, the standard combustion chamber (SCC) and the modified combustion chamber (MCC) geometries were compared in terms of heat and flow in air-cooled, single cylinder DI diesel engines. Numerical analyzes were performed by using AVL Fire ESE Diesel software. The effect of combustion chamber bowl geometries has been researched on temperature-spray droplet distribution, turbulence kinetic energy distributions, turbulence velocity distributions and laminar flame speed distributions parameters.

2. Numerical Simulation

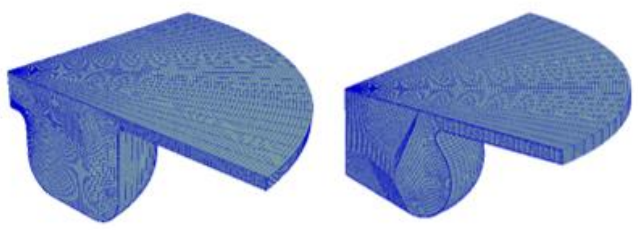
In this study, a numerical study has been carried out in the AVL Fire ESE Diesel section by using the properties of a actual diesel engine. The values for the initial boundary conditions are given in Table 1. In the analysis, 100000 mesh number was determined as the optimum result within the number of 50000, 100000 and 120000 cells and this value was selected. Fig. 1 shows mesh structure used for the combustion chamber simulation in the SCC and MCC piston geometries. It is shown in Fig 2 that the numerical analysis results made with this mesh number are each other close to with the experimental results. These results also show that the accuracy of the numerical analysis is quite high. Also, the models belong to simulation are given in Table 2. The properties and equations used in this modeling are preferred and accepted numerical studies [23].

Table 1. Technical properties and initial boundary conditions of the diesel engine.

Number of cylinders	Single cylender
Type of cooling	Air-cooled
Bore	85 mm
Compression ratio	17.5
Crank radius	42 mm
Number of injection nozzle	4
Injection spray angle	126°
Injection Timing (start and stop)	705° and 729° CA
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)

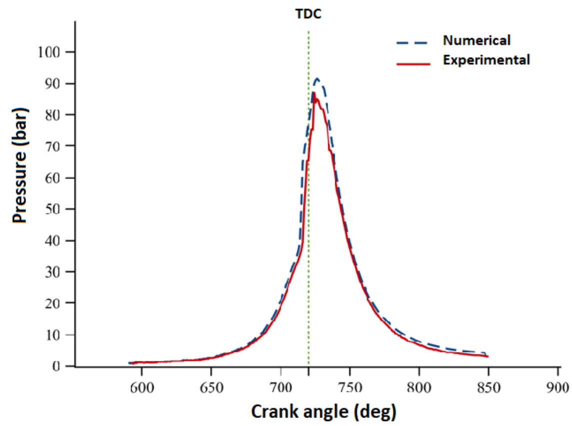
Table 2. Modeling used in numerical analysis

Combustion model	ECFM-3Z
Breakup model	Wave
Turbulence model	K-zeta-f model
Wall interaction model	Walljet1
Evaporation model	Multi component
Soot emission model	Kinetic model

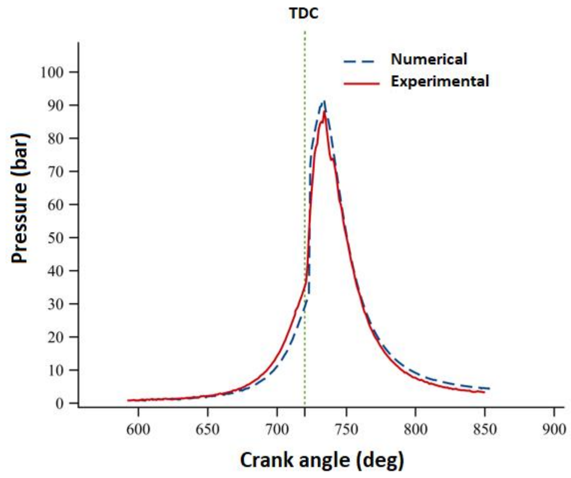


a) b)

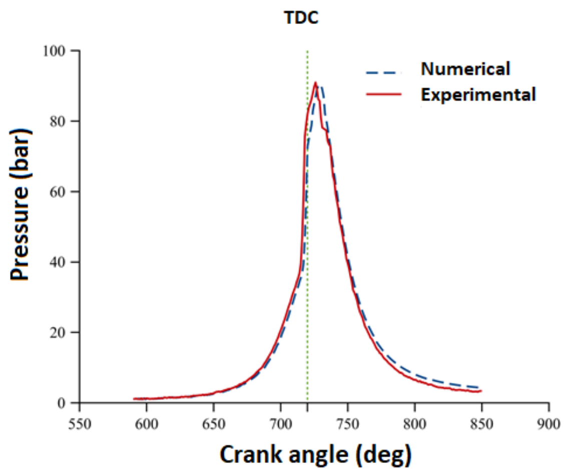
Fig. 1. Computational grid of a) SCC and b) MCC piston bowl geometry at TDC



a)



b)



c)

Fig 2. The change of pressure/crank angle for Diesel (a), 90% Diesel+10 %Canola oil methyl ester (b), 90% Diesel+10 % Sunflower oil methyl ester (c) [23]

3. Results and Discussions

The flow behaviors of Antor 3 LD 510 diesel engine using new design MCC piston geometry was performed and compared with SCC piston geometry. Temperature-spray droplet distribution, flow turbulence kinetic energy distributions and turbulence velocity distribution parameters were examined for flow analysis of chamber geometry. Analyzes were made at 2000 rpm engine speed. The reason for choosing this engine speed is that the maximum torque in the engine is taken at this speed.

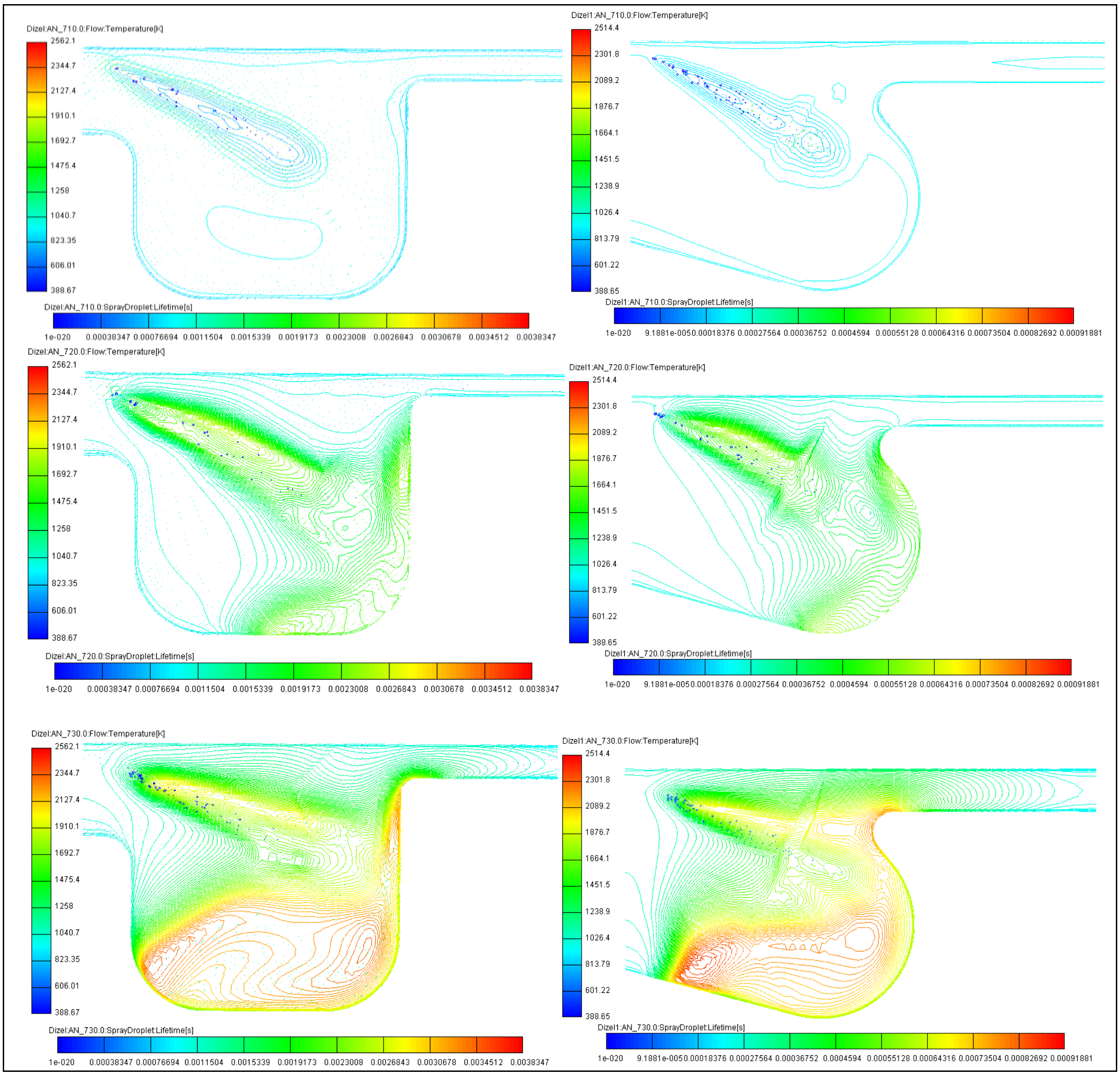


Fig 3. Temperature-spray droplet distribution in SCC and MCC type combustion chamber at 710°, 720° and 730° CA.

First, the temperature of the fuel droplets started to rise from the beginning of the spraying and then the fuel droplets evaporated. As a result of the evaporation of fuel droplets, gases and fuel vapor in-cylinder formed a homogeneous mixture. Eventually, combustion began as a result of the fuel and gas mixture. Fig. 3 shows the instantaneous temperature and spray distribution at certain crank angles belonging to different piston bowl geometries. When the temperature distributions that occur with the mass transfer within the cylinder are examined, it is seen that the combustion is more severe at the 730° CA. It can be said that the combustion chamber temperatures spread over a wider area, especially with the combustion phenomena that develops after the completion of the spraying. It can be said that this situation also affects the fluid temperature. There are many parameters that affect the temperature formation. Many parameters such as mixture formation, flame velocity, turbulence distribution can be said. In the MCC type combustion chamber geometry, as a result of the high penetration depth of the fuel, the wall surface was contacted and evaporated more easily. It can be thought that the decrease in the mean temperature causes the peak temperatures to decrease. For example, when we examine the instantaneous average temperature values in the bowl at 730° CA, this value is seen as 734.95 K in the MCC type combustion chamber, while this value is found as 857.86 K in the SCC type combustion chamber. The instantaneous increase in local temperatures causes many negativities such as increased in-cylinder peak pressures, vibrating operation, increased thermal loads on engine parts and shortened material life. In addition, high temperatures increased turbulence intensity and excessive turbulence intensity is thought to negatively affect combustion.

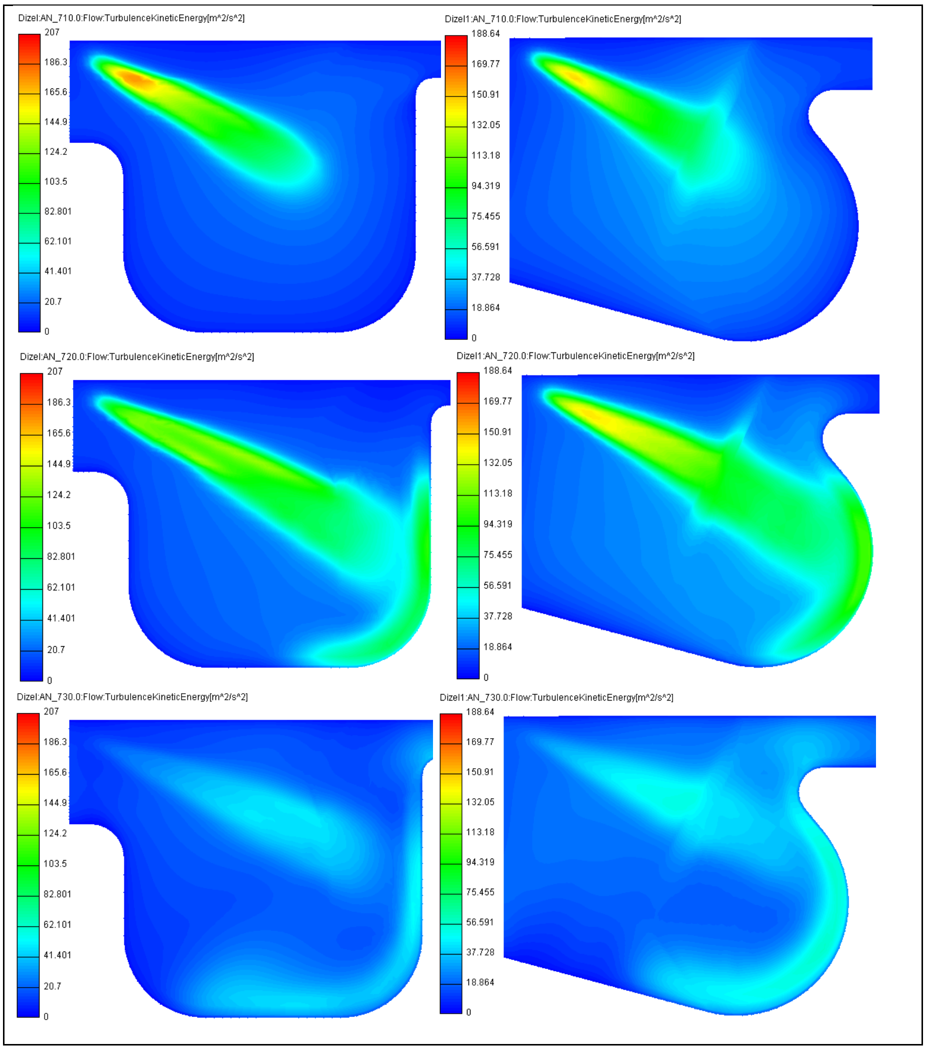


Fig 4. Flow turbulence kinetic energy distributions formed in SCC and MCC combustion chambers

At the compression process, the tumble ‘‘crash’’ caused by the collision of the airflow and the piston top surface enhances the density of turbulent kinetic energy. Fig. 4 shows the change in turbulent kinetic energy belonging to different combustion chambers. Especially, when the distribution formed at three different crank angles during spraying distribution is examined; It is higher in the initial stages of the spray (for 710° CA) compared to other CA’s. This situation shows the effect of the fluid injected into the combustion chamber from the injector. With the advancement of the piston, it is seen that the kinetic energy is distributed over a wider area in both combustion chambers and this depends on the distribution of the fuel. The effect of mass transfer has also decreased with the evaporation of fuel droplets. This situation caused a decrease in turbulent kinetic energy. When the flow of the MCC type chamber geometry is analyzed, it is seen that the fluid impacting the bowl region has a higher turbulent kinetic energy, although the same injection pressure and mass input are equal to the SCC type (Fig. 4). Average turbulent kinetic energy value in the MCC type combustion chamber is 10.53 m^2/s^2, in the SCC type combustion chamber this value is 8.35 m^2/s^2. It can be concluded that the MCC piston geometry can create a strong tumble that contributes to the preparation of the mixture and achieves a high turbulent kinetics that accelerates flame propagation during ignition.

The task of turbulent velocity fields in diesel engines is firstly to prepare the mixture and secondly to control the combustion in large and small scale mixture. Fig. 5 shows the turbulence velocity distributions of both combustion chambers at different crank angles. Also, the values in the turbulence velocity distributions are in parallel with the turbulent kinetic energy change. It is seen that the maximum values reaches and this value decreases with the increase of volume in both combustion chambers at around 720° CA. Similarly, the MCC type compared to the SCC type combustion chamber, the turbulent velocity region was spread in a larger area. It can be said that it is effective in a little increasing the in-cylinder temperatures.

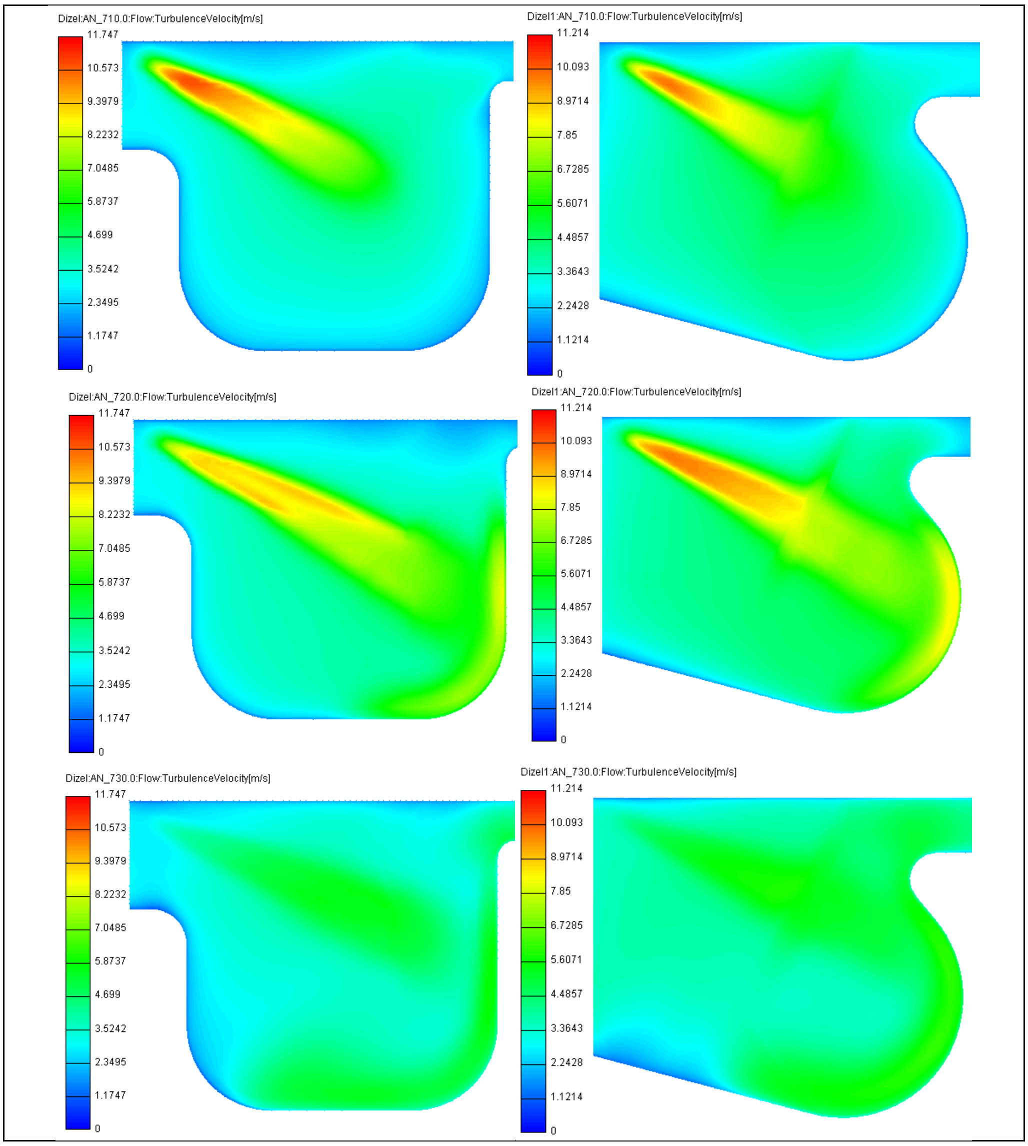


Fig 5. Flow turbulence velocity distributions formed in SCC and MCC combustion chambers

Laminar flame speed is the rate at which the flame propagates into quiescent premixed unburnt mixture in front of the flame. The fuel / air mixture is in a wider area in the area where the fuel is sprayed in the MCC type combustion chamber affects the structure of the flame. Fig. 5 shows the distribution of laminar flame velocities in different combustion chambers. It is seen that the flame is directed towards the center of the combustion chamber, in both combustion chamber geometries. In both models, it is understand that the flame starts in the bowl area and disperses from there into the combustion chamber. The distribution of the flame is of great importance in terms of emissions. As a result of the better mixing of the fuel droplets with the air, the activation energy starts from the wall area of the mixture and its distribution to other regions in the chamber is more pronounced at 720° and 730° CA's. It is seen that the flame extinction zone is less in the MCC type chamber geometry. As is known, the dispersion of the flame in the combustion chamber prevents the increase in local temperatures, and this station contributes significantly to the reduction of thermal loads on engine parts. On the other hand, it is thought that it will make a significant contribution to the reduction of pollutant emission types such as C, CO and HC.

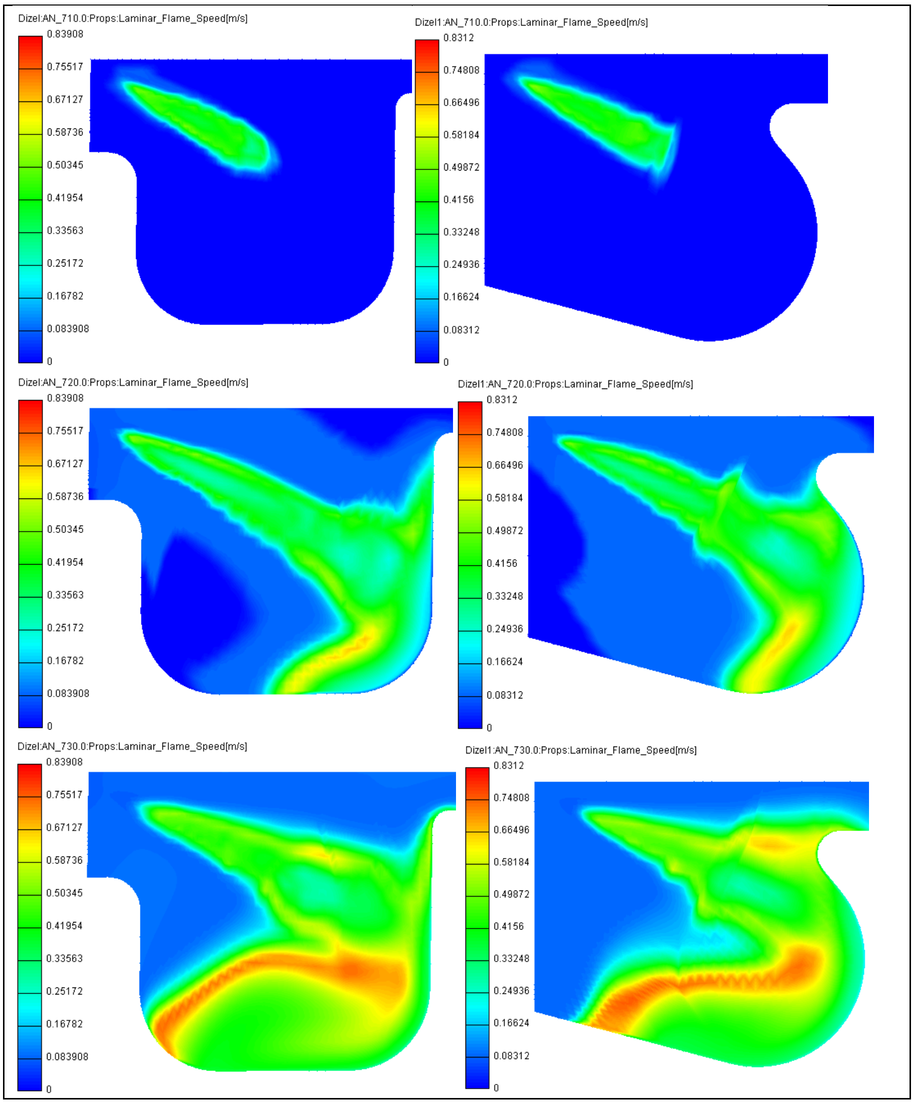


Fig 6. Laminar flame speed distributions formed in SCC and MCC combustion chambers

4. Conclusion

Based upon this study, the standard combustion chamber and the modified combustion chamber bowl geometries were compared by performing heat and flow analysis in the direct injection diesel engines. The 2000 rpm engine speed at which the highest torque was obtained in the engine was selected as the operating condition. Numerical study was performed in AVL Fire ESE Diesel software. The effect of combustion chamber bowl geometry on temperature-spray droplet distribution, turbulence kinetic energy distributions, turbulence velocity distributions and laminar flame speed distributions parameters has been investigated. As a result, when the effect on each parameter is examined, the high penetration depth of the fuel, the wall surface was contacted and evaporated more easily in the modified combustion chamber type chamber geometry. It can be thought that the decrease in the mean temperature causes the peak temperatures to decrease. For example, instantaneous average temperature values in the bowl at 730° CA, this value is seen as 734.95 K in the modified combustion chamber type, while this value is found as 857.86 K in the standard combustion chamber type. Turbulence kinetic energy is higher due to the modified combustion chamber type bowl geometry. Average turbulent kinetic energy value in the modified combustion chamber type was 10.53 m^2/s^2, this value is 8.35 m^2/s^2 in the standard combustion chamber type at 720° CA. The turbulent velocity region was spread in a larger area with the modified combustion chamber type. This situation was increased the laminar flame speed in the modified combustion chamber type.

Acknowledgment

This study was supported by the Scientific and Technological Research Council of Turkey -TÜBİTAK (Project No: 120M143)

Nomenclature

CA	: Crank Angle
CFD	: Computational Fluid Dynamic
DI	: Direct Injection
CA	: Crank Angle
MCC	: Modified Combustion Chamber
SCC	: Standard Combustion Chamber
TDC	: Top Dead Center

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] Demirci A. The effects of different combustion chamber geometries on the performance and emissions of an internal combustion engine. [Istanbul]: Istanbul Technical University; 2017.

[2] Ayaz E, Koten H. Cadirci S. Numerical investigation of cold air flow inside the cylinder of a heavy duty diesel engine. International Journal of Advances in Engineering and Pure Sciences. 2018; 30(4): 445-52.

[3] Yilmaz M. Comparison of ethanol and gasoline in engine cylinder flow. [Bursa]: Bursa Uludag University; 2019.

[4] Abay K, Colak U, Yuksek L. Computational fluid dynamics analysis of flow and combustion of a diesel engine. Journal of Thermal Engineering. 2018; 4(2): 1878-95.

[5] Monaghan MN, Pettifer HF. Air motion and its effect on diesel performance and emissions. SAE Technical Paper. 1981; 810255: 19-35.

[6] Shimoda M, Shigemori M, Tsuruoka S. Effect of combustion chamber configuration on in-cylinder air motion and combustion characteristics of D.I. Diesel engine. SAE Technical Paper. 1985; 850070: 1-13.

[7] Buyukkaya E. Flow modeling in a DI Diesel engine combustion chamber using CFD. Fuels, Fire and Combustion in Engineering Journal. 2016; 2: 31-8.

[8] 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. 2019; 8(2): 83-102.

[9] Stephenson PW, Claybaker PJ, Rutland CJ. Modeling the Effects of intake generated turbulence and resolved flow structures on combustion in DI Diesel engines. SAE Technical Paper. 1996; 960634: 103-17.

[10] Gunabalan A, Ramprabhu R. Effect of piston bowl geometry on flow, combustion and emission in DI engines-a CFD approach. International Journal of Applied Engineering Research. 2009; 4(11): 2181–88.

[11] Zheng Z, Liu C, Tian X, Zhang X. Numerical study of the effect of piston top contour on GDI engine performance under catalyst heating mode. Fuel. 2015; 157: 64-72.

[12] Chu S, Shin H, Kim K, Moon S, Min K, Jung H, Kim H, Chi Y. An experimental investigation of in-cylinder flow motion effect on dual-fuel premixed compression ignition characteristics. SAE Technical Paper. 2020; 2020-01-0306: 1-12.

[13] Heywood JB. Internal Combustion Engine Fundamentals. United States: McGraw-Hill Education; 2018.

[14] Perini F, Zha K, Busch S, Kurtz E, Peterson RC, Warey A, Reitz RD. Piston geometry effects in a light-duty, swirl-supported diesel engine: Flow structure characterization. International Journal of Engine Research. 2018; 19(10): 1079-98.

[15] Khan S, Panua R, Bose PK. Combined effects of piston bowl geometry and spray pattern on mixing, combustion and emissions of a diesel engine: A numerical approach. Fuel. 2018; 225: 203-17.

[16] Sener R, Ozdemir MR, Yangaz MU. Influence of piston bowl geometry on combustion and emission characteristics. Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy. 2019; 233(5): 576–87.

[17] Li J, Yang WM, An H, Maghbouli A, Chou SK. Effects of piston bowl geometry on combustion and emission characteristics of biodiesel fueled diesel engines. Fuel. 2014; 120: 66-73.

[18] Mohan B, Yang W, Yu W, Tay KL, Chou SK. Numerical investigation on the effects of injection rate shaping on combustion and emission characteristics of biodiesel fueled CI engine. Applied Energy. 2015; 160: 737-45.

[19] Aksoy F, Yilmaz E. An investigation of combustion and performance characteristics of a direct injection diesel engine fuelled with the blends of 10% fish oil biodiesel and 90% diesel fuel. Gazi University Journal of Science Part C: Design and Technology. 2019; 7(1): 12-24.

[20] Lešnik L, Kegl B, Torres-Jiménez E, Cruz-Peragón F, Mata C, Biluš I. Effect of the in-cylinder back pressure on the injection process and fuel flow characteristics in a common-rail diesel injector using GTL fuel. Energies. 2021; 14(452): 1-21.

[21] Bishop D, Situ R, Brown R, Surawski N. Numerical modelling of biodiesel blends in a diesel engine. Energy Procedia. 2017; 110: 402-7.

[22] Hasan AAKM, Mashkour MA, Mohammed AA. Burning velocity measurement of biodiesel fuel and its blends using particle imaging path technique and image processing. Journal of Engineering and Sustainable Development. 2018; 22(2): 1-17.

[23] Temizer I, Eskici B. Investigation on the combustion characteristics and lubrication of biodiesel and diesel fuel used in a diesel engine. Fuel. 2020; 278: 1-9.