e-ISSN: 2146 - 9067

International Journal of Automotive Engineering and Technologies

journal homepage: http://ijaet.academicpaper.org

Original Research Article

A Comparative Analysis of In-Cylinder Flow, Heat Transfer and Turbulence Characteristics in Different Type Combustion Chamber

Müjdat Fırat[1,*], Yasin Varol[2]

[1]Fırat University, Faculty of Technical Education, Department of Mechanical Education, Elazığ, Turkey

[2]Fırat University, Faculty of Technology, Department of Automotive Engineering, Elazığ, Turkey

ARTICLE INFO

* Corresponding author
mujdatfirat@gmail.com

Received: June 13, 2017
Accepted: February 26, 2018

Published by Editorial Board Members of IJAET

ABSTRACT

Recently, some studies have been carried out on the combustion chamber design for diesel and gasoline engines; however, the piston bowl model approach which may neglect interactions with cylinder head design have routinely used**.** Fluid motion within the cylinder of internal combustion engines has a significant influence on the combustion process and engine efficiency. The aim of this study, two different geometries of the combustion chamber of internal combustion engine were compared in terms of in-cylinder flow and heat transfer during the intake process. For this, complete calculations of the intake stroke were performed under realistic operating conditions and heat transfer, velocity and turbulence flow fields obtained from each combustion chamber investigation in detail. The data were solved by finite volume method and ANSYS-Fluent 12.0 commercial code. The findings include using of the k-ε turbulence model and dynamic mesh generation model for different crank angles. The results showed that flow structure and turbulence intensity were affected by combustion chamber geometries. Also, the pent-roof type combustion chamber configuration was found to generate turbulence intensity more efficiently than the conventional type combustion chamber.

Keywords: Turbulence Intensity, Combustion Chamber, Dynamic mesh model, CFD

1. Introduction

In recent years, internal combustion engines have been greatly improved in terms of efficiency, emissions and performance. The current trend, the improvement and the development of sub-components of the engine which are injection phenomena, intake system, exhaust systems and combustion chamber design parameters [1-4]. In addition, the combustion process depends properly on fuel-air mixture. In-cylinder flow and fuel-air mixture are determinant of combustion in internal combustion engines which provide significant effects on the engine efficiency and emissions. The in-cylinder flow characteristics have been investigated both numerically and experimentally depending on the shape of the combustion chamber by some researchers. For the investigation of in-cylinder flow, a wide range of optical measurement techniques have been developed in recent years. So, it is possible to determine individual droplet size and velocities by means of Laser Doppler Anemometry (LDA), Laser Doppler Velocimetry (LDV), Particle Image Velocimetry (PIV), Raman Spectroscopy etc. by experimental method [5-9]. Müller et al. carried out an experimental study to measure in an optical accessible, direct injection spray-guided internal combustion engine using of the high-speed PIV. Flow structure behaviors such as main vortex center, flow field and kinetic energy were measured by means of optical technique [10]. Moreover, a similar study performed on realistic engine speeds as experimentally by Stansfield et al. [11]. Kang and Baek [12] were experimentally investigated turbulence characteristics of the tumble flow in a four valve engine by LDV and analyzed turbulence intensity, length scales, velocity and energy spectrum. However, there have been numerous computational techniques on in-cylinder flow in internal combustion engines. In this field, there are many computational commercial (ANSYS-Fluent, KIVA, FIRE, STAR-CD etc.) or personal codes by using finite volume and finite element for numerical calculation [13]. In this content, Payri et al. [14] studied the three dimensional calculation of the intake and compression stroke of a four- valve direct injection diesel engine. Flow structure was affected by different combustion chamber and piston bowl models predicted by the computational fluid dynamics (CFD). Johan et al. [15] simulated in-cylinder cold flow using a finite element method. Their results were presented for exhaust, intake and compression strokes of the realistic cylinder operations. Banaeizadeh at al. [16] conducted on a computational study to show Large-eddy simulations of turbulent flows in internal combustion engines. The flow statistics predicted by the LES model have shown to compare with the available experimental data. The effects of engine parameter such as mean piston speed, crank angle and cycles on in-cylinder flow mean axial velocity, vorticity magnitude, turbulent viscosity, fuel droplet distribution etc. were studied. Bilgin [17] performed a numerical simulation of the in-cylinder flow in an axisymmetric non compressing engine like geometry. The results were shown in terms of velocity vectors, stream lines, axial velocity and turbulence intensity. Also, he compared the obtained results with measurements in the literature. Mohammadi and Yaghouni [18] were computed instantaneous local heat transfer coefficient on the combustion chamber in a spark ignition engine by using CFD code. Their study was carried out with different engine speeds and the results were shown in terms of Nusselt and Reynolds number.

The main aim of this comparative study is to investigate the heat transfer, fluid flow and turbulence structure in a conventional and pent-roof type combustion chamber and effects of the combustion chamber shape on the flow and turbulence intensity during intake stroke. Also, the study shows how flow acts inside of whole intake cycle and it clarifies some places in which designer should consider some geometrical innovations.

2. Governing Equations

Calculation of the temperature, turbulence and flow field in a combustion chamber of internal combustion engine requires to obtain the solution of the governing equations. The compressible, unsteady and turbulent in-cylinder flow can be described by differential equations of continuity, momentum, energy, turbulence kinetic energy and its dissipation rate. Radiation mode of heat transfer is neglected according to other modes of heat transfer. Buoyancy forces are also neglected and heat transfer regime is accepted as forced convection. The mass conservation equation can be written as the following:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho V) = 0 \tag{1}$$

Momentum equations can be written in two directions as:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = -\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(\lambda\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)V + 2\mu\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] \tag{2}$$

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho uv)}{\partial x} = -\frac{\partial p}{\partial y} + \frac{\partial}{\partial x}\left[\mu\left(\frac{\partial v}{\partial x} + \frac{\partial u}{\partial y}\right)\right] + \frac{\partial}{\partial y}\left(\lambda\nabla V + 2\mu\frac{\partial v}{\partial x}\right) \tag{3}$$

Energy equation:

$$\frac{\partial(\rho e)}{\partial t}+\nabla(\rho e V)=\frac{\partial}{\partial x}\left(k\frac{\partial T}{\partial x}\right)+\frac{\partial}{\partial y}\left(k\frac{\partial T}{\partial y}\right) \tag{4}$$

The turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu_{eff}+\frac{\mu_t}{\sigma k}\right)\frac{\partial k}{\partial x_i}\right]+Gk+Gb-\rho\varepsilon-Y_M \tag{5}$$

$$\rho\frac{De}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu_{eff}+\frac{\mu_t}{\sigma\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{\varepsilon 1}\frac{\varepsilon}{k}(Gk+C_{\varepsilon 3}Gb)-C_{\varepsilon 2}\rho\frac{\varepsilon^2}{k} \tag{6}$$

There are numerous alternative turbulence modeling approaches of varying degree of complexity, but in this work, k-ε turbulence model was used to forecast the flow in the cylinder of an incompressible fluid [19]. In k-ε model, the turbulent or eddy viscosity concept, and calculation of turbulent viscosity μ_t according to Prandtl-Kolmogorov relation is given as the following [20]:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon} \tag{7}$$

$C_{\varepsilon 1}$, $C_{\varepsilon 2}$ and $C_{\varepsilon 3}$ are model constant. These coefficients are,

$C_{\varepsilon 1}$ = 1.44; $C_{\varepsilon 2}$ = 1.92; $C\mu$ = 0.09; μ_k = 1.0; σ_ε = 1.3

Total effective viscosity of the flow is then, given by the combination of the turbulent viscosity and laminar viscosity as:

$$\mu_{eff}=\mu_t+\mu \tag{8}$$

2.1. Boundary Conditions

The inlet boundary conditions were obtained from calculated instantaneous mass flow rate. This can be done due to acceptation of the incompressibility. Also, no-slip boundary conditions were applied for all velocities at walls. The two dimensional Navier-Stokes equations are solved on a moving mesh and turbulent fluxes are modeled by using k-ε model. Boundary conditions for the considered physical model are given as the following [3,23]:

Inlet Temperature, T_{inlet} = 310 K

Side temperature of combustion chamber, T_{side} = 493 K

Temperature of Piston Head, T_{piston} = 493 K

Inlet velocities, V=33 m/s (relative to average piston speed) for n= 3000 rpm

Velocities at side of cylinder, u = 0, v = 0

Velocities at piston surface, u = 0, v = v_{piston} .

3. Numerical Procedure

This study investigated the effects of the combustion chamber shape on the turbulence and flow field and calculation of the intake stroke. To do this, two different combustion chamber geometries were analyzed. In-cylinder flow has been simulated using ANSYS-Fluent commercial code [20]. This code uses finite volume method in order to solve Navier – Stokes and energy equations and it is widely used in the field of internal combustion engines design. The finite volume method can accommodate any type of grid. Thus, it is suitable for complex geometries, like present study. The standard k-ε model is used for all analysis. The computational code is based on the pressure-correction method and uses the SIMPLE algorithm [21]. The first order upwind differencing scheme is used for momentum, energy and turbulence equations. The dynamics grid approach is used to treat the moving piston in the computational area. The calculations are started with a crank angle of top dead center (TDC) and finished bottom dead center (BDC) for at engine speed as 3000 rpm. The model structure is a hybrid grid and to setup boundary condition for moving piston. The engines specifications used in the computations are listed in Table 1.

Table 1. Engines geometry and parameters

	Case I	Case II
Bore / mm	80	80
Stroke / mm	90	90
Intake valve angle /deg	0	20
Crank period / deg	180	180
Engine speed /rpm	3000	3000

The piston at top dead center when the crank angle is assumed to be zero and the calculations started and continued during 180 crank angle. In addition, psychical models and measurement section of axial velocity and turbulence intensity shown in Fig. 1.
It is consisting of realistic engine geometry which are used in two-dimensional studies with their geometric values. The obtained axial velocity values were compared to other known values reported by other researchers as shown in Fig. 2. Therefore, the converged value agrees very well with other values obtained in the literature [22]. Comparison was performed in the same area within the cylinder.

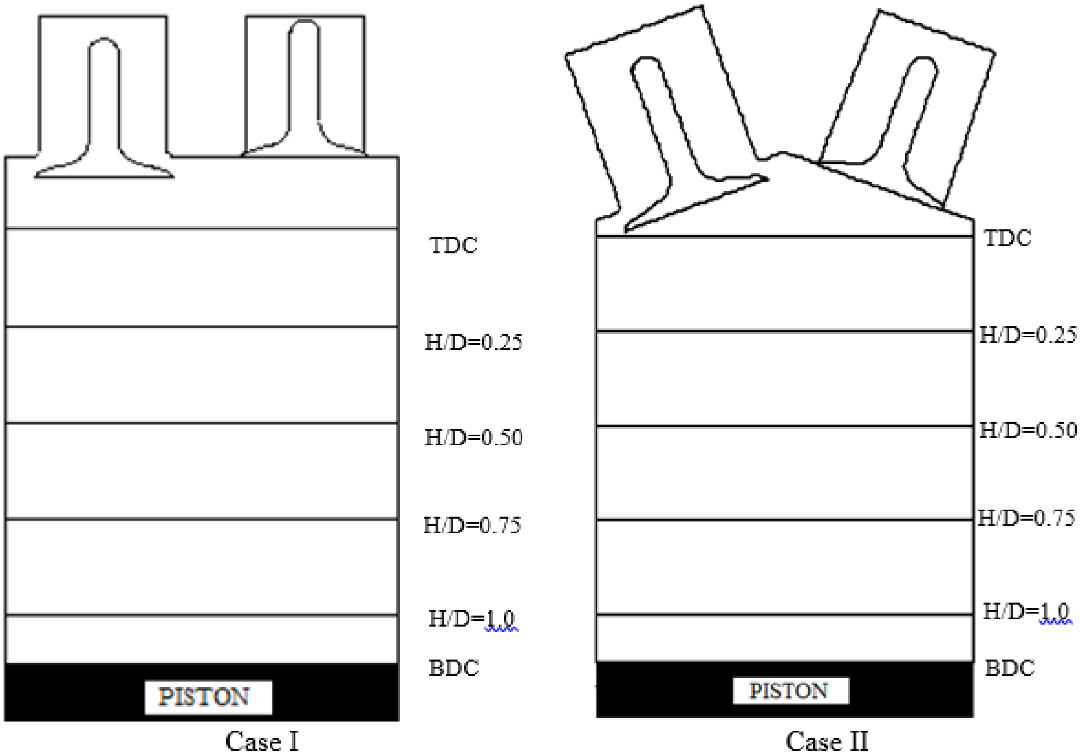


Figure 1. Measurement section for Case I and Case II

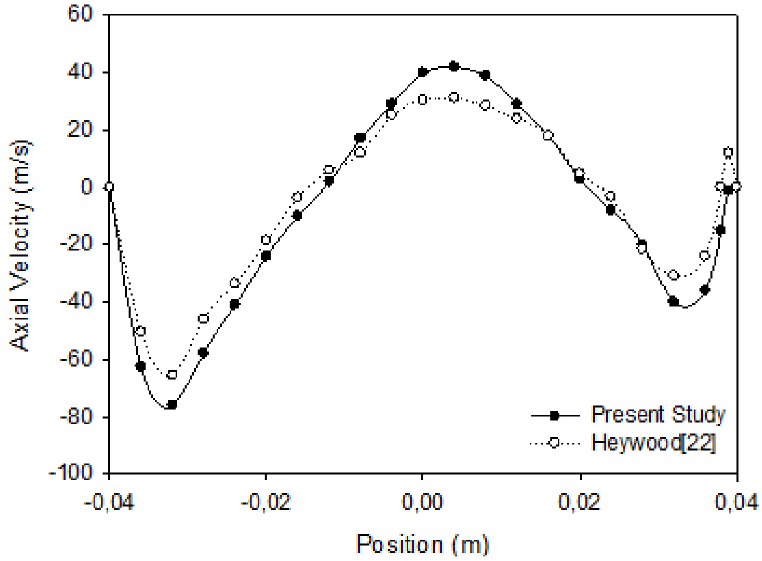


Figure 2. Comparison of obtained axial velocity with the literature

4. Results and Discussion

A numerical analysis has been performed in this paper for different crank angle and combustion chamber. The results were presented with streamlines, velocity vector, isotherms, axial velocity and turbulence intensity. As presented above, the standard k–ε model and single intake port were carried out for all two-dimensional models [23].

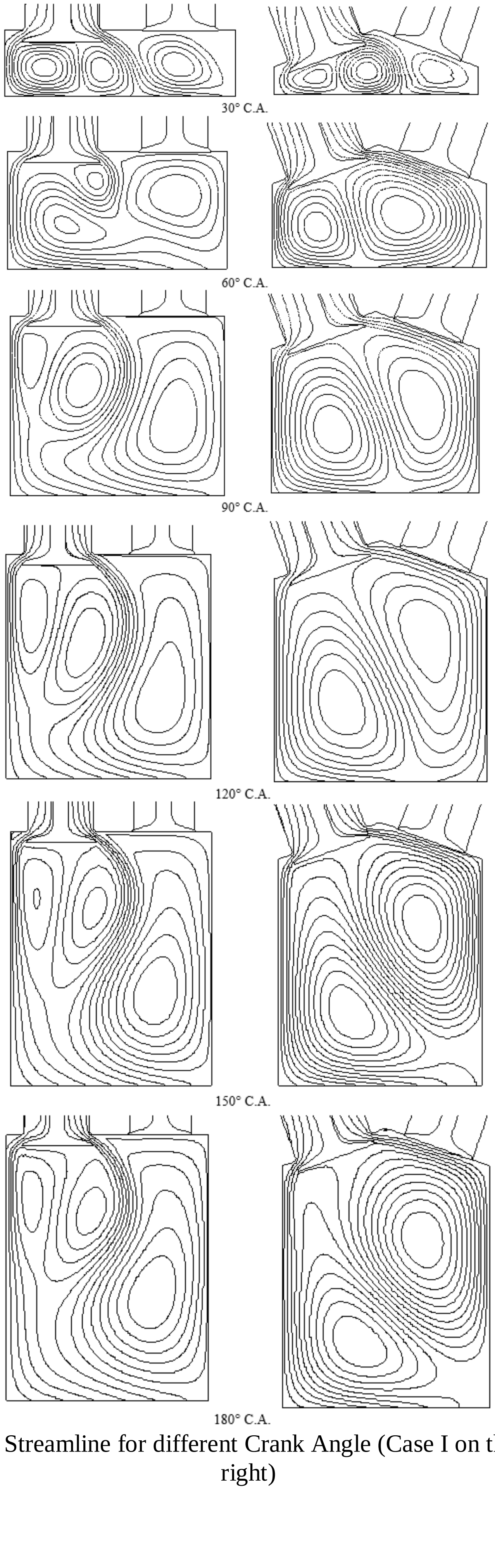
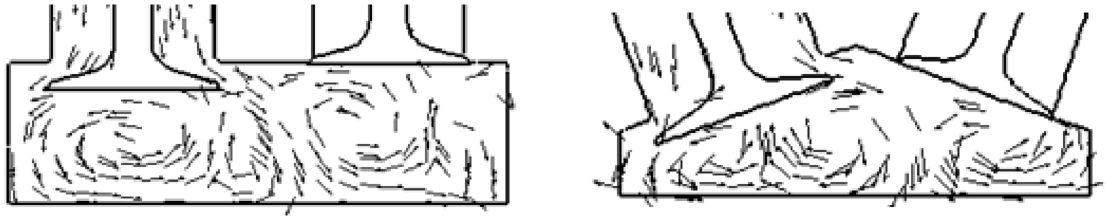
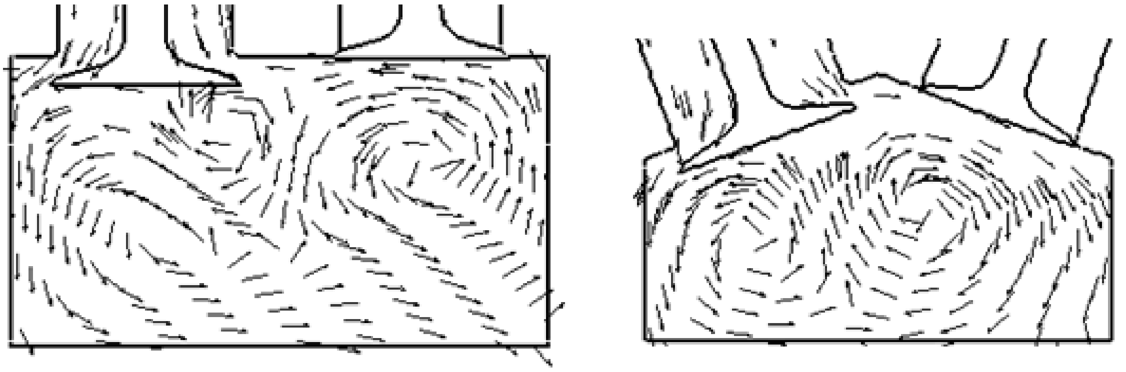


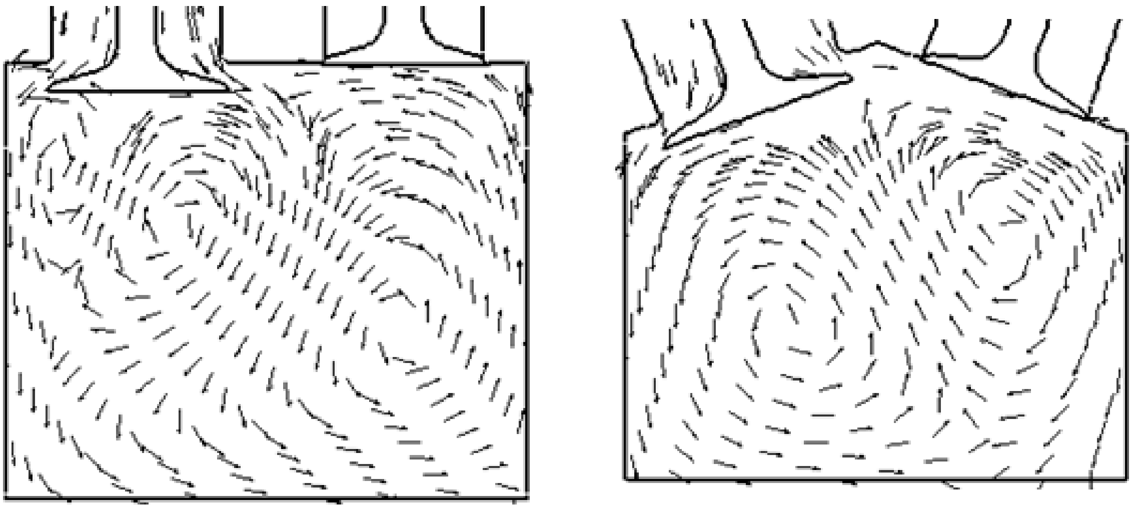
Figure 3. Comparison of Streamline for different Crank Angle (Case I on the left and Case II on the right)



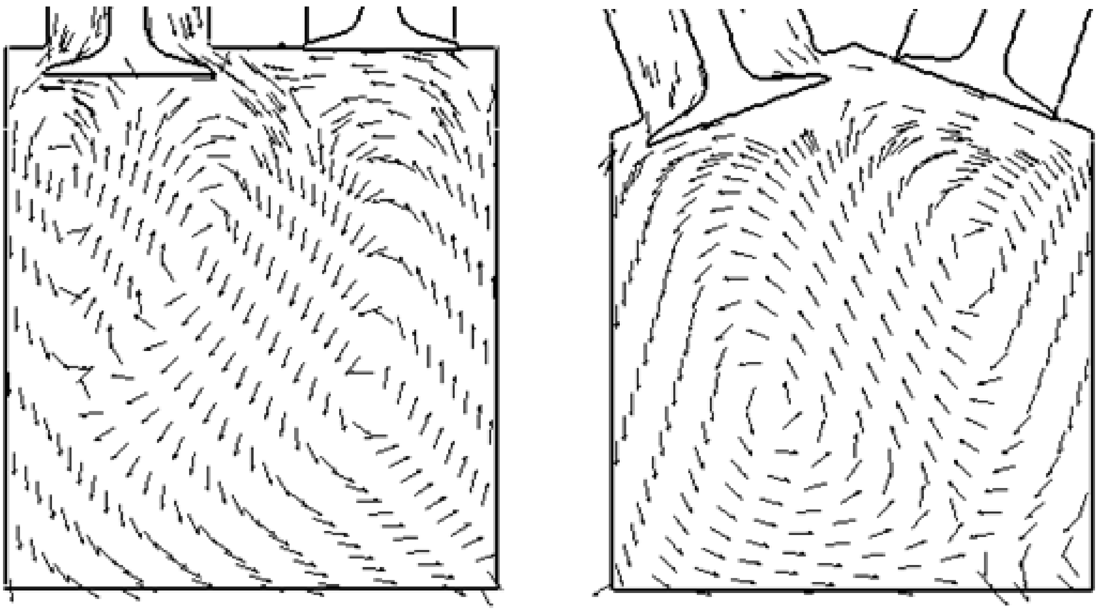
30° C.A.



60° C.A.



90° C.A.



120° C.A.

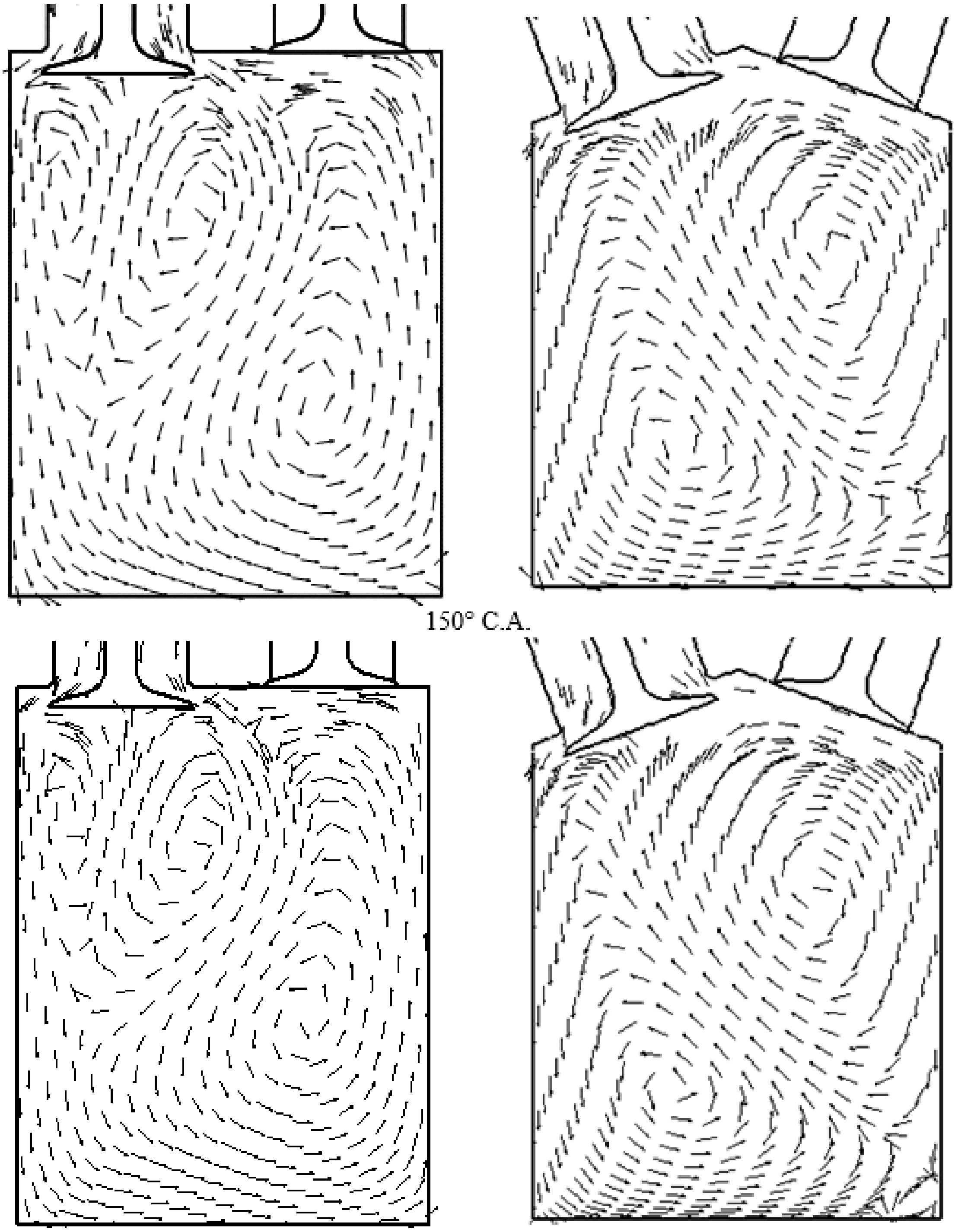
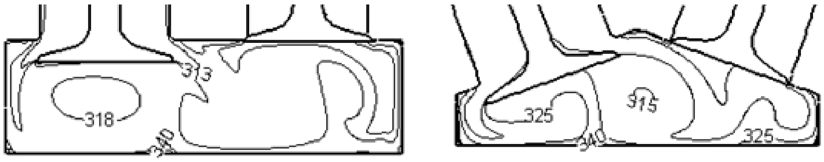
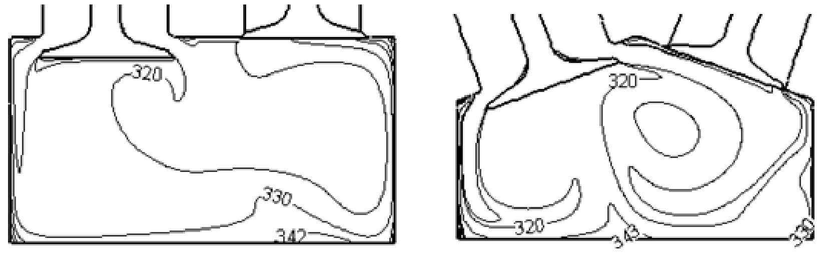


Figure 4. Comparison of velocity vector for different crank angle (Case I on the left and Case II on the right)

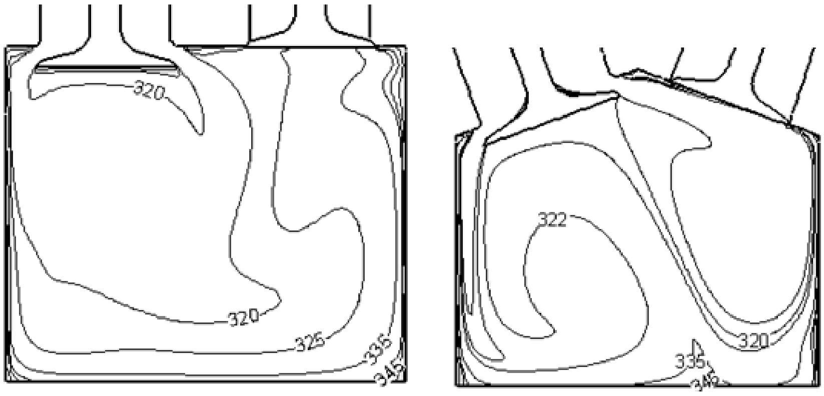
Fig. 5 shows the isotherms of same crank angle of Fig. 3. It was noticed that temperature boundary conditions for combustion chamber was measured from realistic engine. This case, constant temperature boundary conditions were kept as 493 K. It is clearly seen from the isotherms that the temperature of fluid inside the combustion chamber increases with increasing of crank angle for case I and Case II. In this case depends on the higher in-cylinder flow velocity and sweep out inside of the cylinder. Pent-roof type combustion chamber inside cylinder temperature higher than conventional type combustion chamber as presented in Fig. 5. This result indicates more efficient of pent-roof type combustion chamber (Case II) in terms of in-cylinder heat transfer. Nevertheless, the crank angle and combustion chamber type are effective parameters on this parameter.



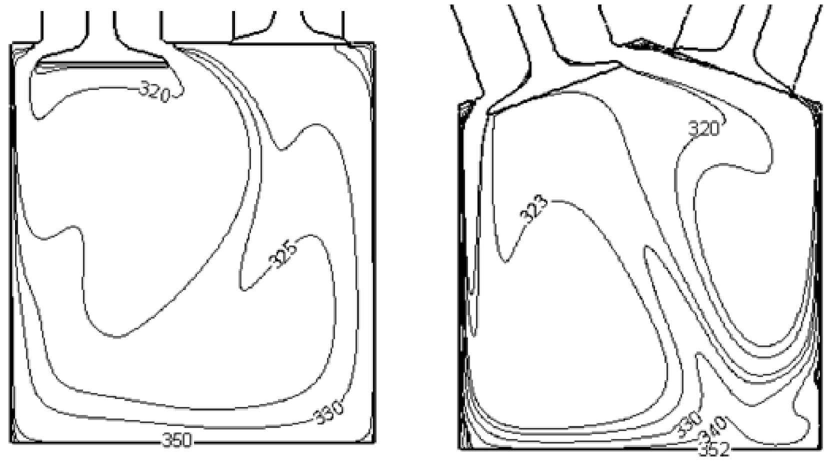
30° C.A.



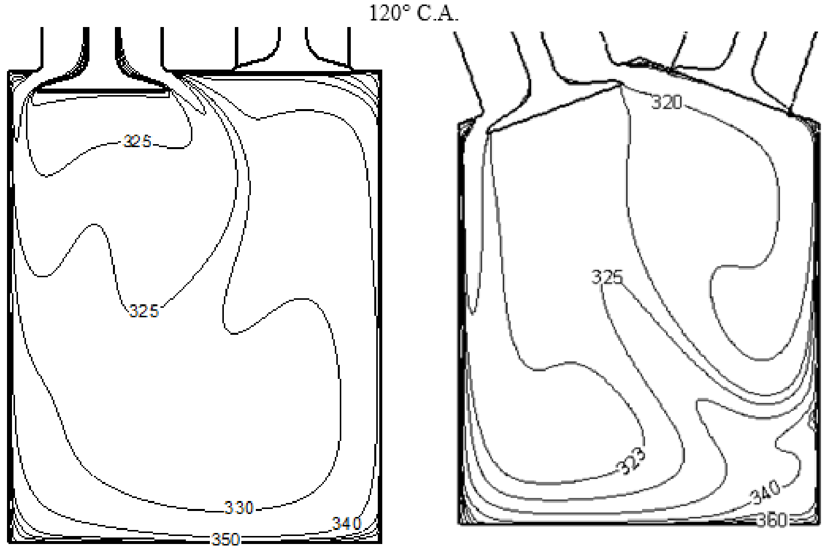
60° C.A.



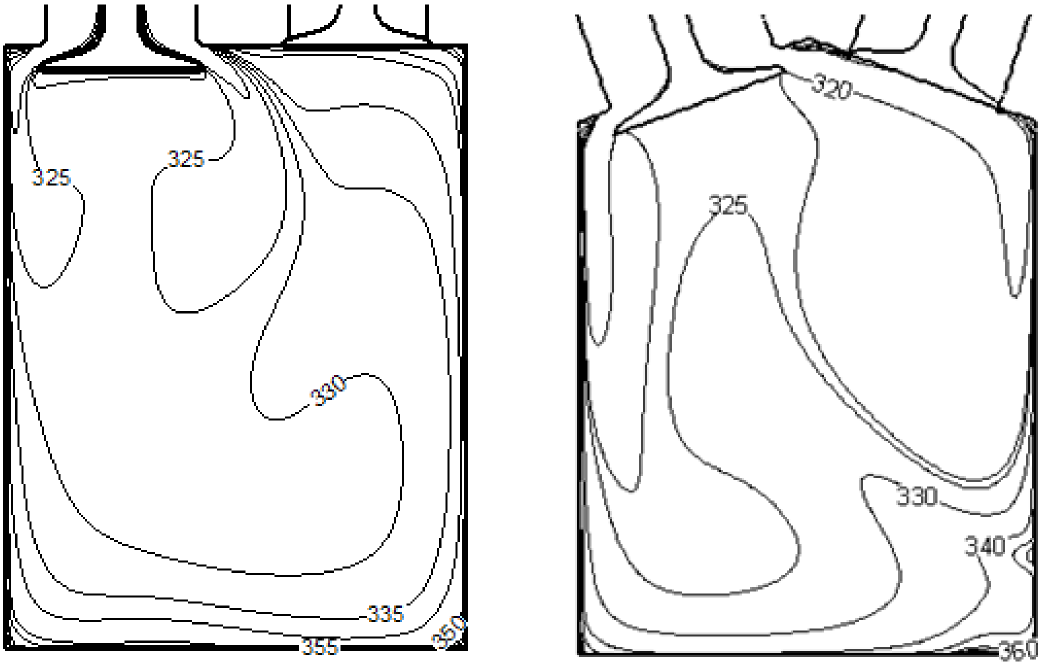
90° C.A.



120° C.A.



150° C.A.



180° C.A.

Figure 5. Comparison of isotherms for different Crank Angle (Case I on the left and Case II on the right)

Fig. 6 and Fig. 7 show the axial velocity profiles and turbulence intensity variation at five different locations on the cross-sections for the two different combustion chamber model (Case I and Case II, respectively). It is clearly seen the effects of intake valve in the axial velocity distribution as shown in Fig. 6(a) and Fig. 7(a). Due to the front of intake valve, the axial velocity near the wall are higher than those in the pent-roof type combustion chamber case, which leads to higher axial flows under the intake valve side of upper cylinder. In the realistic engine configuration, this non-uniform distribution of the axial velocity even at the piston position (H/D=1.0) could cause variations of turbulence intensity. The axial velocity is generated significantly in the upper part of the cylinder. Due to the non-uniform distribution of the mean flow velocity in the lower part of cylinder, the turbulence intensity is higher near the cylinder walls. However, turbulence intensity decreases to the exhaust valve side and bottom of the cylinder decreases as presented in Figs. 6b and 7b. Nevertheless, the axial velocity and turbulence intensity in the Case II higher than Case I as shown in Fig. 6 and Fig. 7.

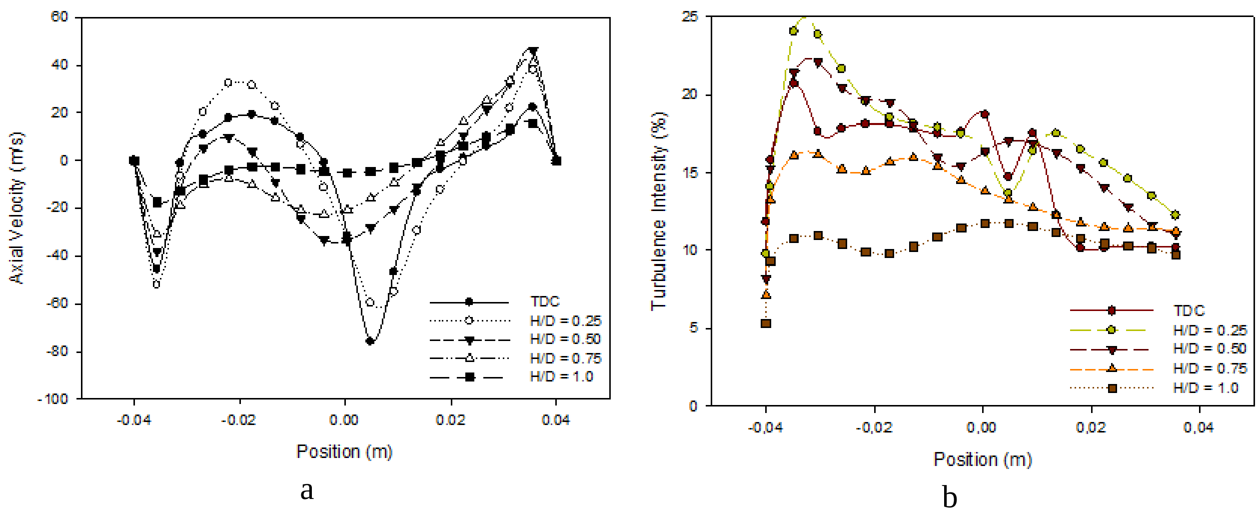
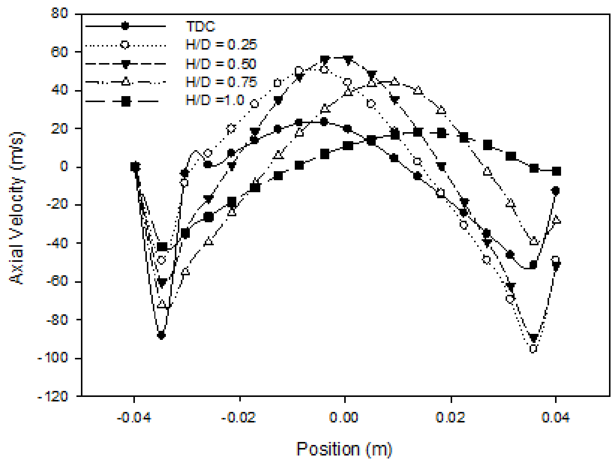
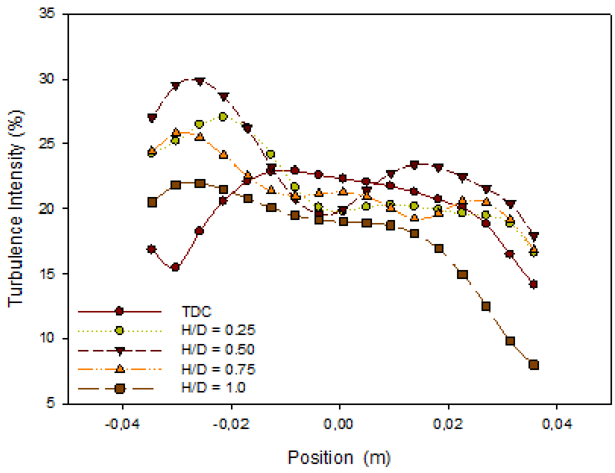


Figure 6. a) Axial velocity, b) Turbulence Intensity for Case I



a



b

Figure 7. a) Axial velocity, b) Turbulence Intensity for Case II

5. Conclusions

In this study, two different geometries of the combustion chamber of internal combustion engine were compared in terms of in-cylinder flow and heat transfer during the intake process. To do this, complete calculations of the intake stroke were performed under realistic operating conditions and heat transfer, velocity and turbulence flow fields obtained from each combustion chamber investigation in detail. The data were solved by finite volume method and ANSYS-Fluent 12.0 commercial code. The findings include using of the k-ε turbulence model and dynamic mesh generation model for different crank angles. The obtained results can be summarized as the following:

- The type of combustion chamber in internal combustion engines can be control parameter for heat transfer and fluid flow.
- Temperature of in-cylinder fluid increases with increasing of crank angle. In addition, pent-roof type combustion chamber inside cylinder temperature higher than conventional type combustion chamber.
- The pent-roof type combustion chamber configuration was found to generate turbulence intensity more efficiently than the conventional type combustion chamber.

Turbulence intensity and axial velocity are higher near the cylinder walls of the intake valve side. Turbulence intensity is particularly important during the intake stroke. The turbulence intensity is very significant in terms of the formation of the mixture and the performance of the engine.

Acknowledgements

Authors thank to Firat University Scientific and Research Fund for their valuable financial supported with projects number 1874 and TEKF 12.06.

Nomenclature

Gk The generation of turbulence kinetic energy due to the mean velocity gradients
Gb The generation of turbulence kinetic energy due to buoyancy
E Internal energy per unit mass (J/kg)
K Thermal conductivity (W/mK)
N Engine speed (rpm)
P Pressure (Pa)
t Time (s)
T Temperature (K)
u, v, w Velocity magnitudes in direction x,y,z (m/s)
V Volume (m^3)
Vc Combustion chamber volume
Greek symbols
σε The turbulent Prandtl numbers for ε
μ Dynamic viscosity (Pa /s)
μt Turbulence viscosity
Ө Crank angle
Ε Turbulent dissipation rate (m^2 / s^3)
ρ Fluid density (kg/m3)
σk The turbulent Prandtl numbers for k

6. References

[1] Gold M. R., Arcoumanis C., Whitelaw J. H., “Mixture preparation strategies in an optical four valve port-injected gasoline

engine", Int. J. Engine Research, 1, 41-56, 2000.

[2] Song J., Yao C., Liu Y., Jiang Z., Investigation on flow field in simplified piston bowls for DI diesel engine, Eng. Appl. Comp. Fluid Mech., 2, 354-365, 2008.

[3] Varol Y., Oztop H. F., Firat M., Koca A., CFD modeling of heat transfer and fluid flow inside a pent-roof type combustion chamber using dynamic model, Int. Comm. Heat Mass Trans., 37, 1366-1375, 2010.

[4] Jemni M. A., Kantchev G., Abid M. S., Influence of intake manifold design on in-cylinder flow and engine performances in a bus diesel engine converted to LPG gas fuelled, using CFD analyses and experimental investigations, Energy, 36, 2701-2715, 2011.

[5] Keskinen J.P., Vuorinen V., Kaario O., Larmi M., Large eddy simulation of a piston–cylinder assembly: The sensitivity of the in-cylinder flow field for residual intake and in-cylinder velocity structures, Computers and Fluids 122, 123–135, 2015.

[6] Fajardo C., Sick V., Flow field assessment in a fired spray-guided spark-ignition direct injection engine based on UV particle image velocimetry with sub crank angle resolution, Proc. Comb. Institute, 31, 3023-3031, 2007.

[7] Kang K. Y., Reitz R. D., The effect of intake valve alignment on swirl generation in a DI diesel engine, Exp. Ther.Fluid Sci., 20, 94-103, 1999.

[8] Egerman J., Koebcke W., Ipp W., Leipertz A., Investigation of the mixture formation inside a gasoline direct injection engine by means of linear raman spectroscopy, Proc. Comb. Institute, 28, 1145-1152, 2000.

[9] Wu C., Deng K., Wang Z., The effect of combustion chamber shape on cylinder flow and lean combustion process in a large bore spark-ignition CNG engine, Journal of the Energy Institute, 89, 240-247, 2016.

[10] Müller S. H. R., Böhm B., Gleibner M., Grzeszik R., Arndt S., Dreizler A., Flow field measurements in an optical accessible, direct injection spray-guided internal combustion engine using high-speed PIV, Exp. Fluids, 48, 281-290, 2010.

[11] Stanfield P., Wigley G., Justham T., Catto J., Pitcher G., PIV analysis of in-cylinder flow structures over range of realistic engine speeds, Exp. Fluids, 43, 135-146, 2007.

[12] Kang K. Y., Baek J. H., Turbulence characteristics of tumble flow in a four-valve engine, Exp. Ther. Fluid Sci., 18, 231-243,1998.

[13] Basha S. A., Gopal K. R., In-cylinder fluid flow, turbulence and spray models-A review, Ren. Sust. Energy Reviews, 13, 1620-1627, 2009.

[14] Payri F., Benajes J., Margot X., Gil A., CFD modeling of the in-cylinder flow in direct-injection diesel engine, Computer & Fluids, 33, 995-1021, 2004.

[15] Johan Z., Moraes A. C. M., Buell J. C., Ferencz R. M., In-cylinder cold flow simulation using a finite element method, Comp. Methods Appl. Mech. Eng., 190, 3069-3080, 2001.

[16] Banaeizadeh A., Afshari A., Schock H., Jaberi F., Large-eddy simulations of turbulent flows in internal combustion engines, Int. J. Heat Mass Trans., 60, 781–796, 2013.

[17] Bilgin A., Numerical Simulation of the cold flow in an axisymmetric non-compressing engine-like geometry, Int. J. Energy Res., 23, 899-908, 1999.

[18] Mohammadi A., Yaghoubi M., Estimation of instantaneous local heat transfer coefficient in spark-ignition engines, Int. J. Thermal Sci., 49, 1309-1317, 2010.

[19] Launder B.E., Spalding D.B., Lectures in Mathematical Models of Turbulence, Academic Press, London, 1972.

[20] ANSYS-Fluent 12.0, User Guide, 2009.

[21] Patankar S. V., Numerical heat transfer and fluid flow, Hemisphere, New York, 1980.

[22] Heywood J. B., Internal Combustion Engine Fundamentals, McGraw-Hill, pp.809, 1998.

[23] M. Firat, Modeling of fluid flow and heat transfer in new generation combustion chambers, MSc Thesis, Firat University, Inst. Science Graduate School Natural Applied Sciences, 2010.