Numerical Simulation of Flow and Combustion in an Axisymmetric Internal Combustion Engine

Nureddin Dinler, and Nuri Yucel

Abstract**—**Improving the performance of internal combustion engines is one of the major concerns of researchers. Experimental studies are more expensive than computational studies. Also using computational techniques allows one to obtain all the required data for the cylinder, some of which could not be measured. In this study, an axisymmetric homogeneous charged spark ignition engine was modeled. Fluid motion and combustion process were investigated numerically. Turbulent flow conditions were considered. Standard k-ε turbulence model for fluid flow and eddy break-up model for turbulent combustion were utilized. The effects of valve angle on the fluid flow and combustion are analyzed for constant air/fuel and compression ratios. It is found that, velocities and strength of tumble increases in-cylinder flow and due to increase in turbulence strength, the flame propagation is faster for small valve angles.

Keywords**—**CFD simulation, eddy break-up model, k-ε turbulence model, reciprocating engine flow and combustion.

I. Introduction

UNDERSTANDING the nature of the flows and combustion in internal combustion engines are important for improving engine performance. The flows in IC engines can be characterized by swirl, tumble and compression in the cylinder. This flow motion has a strong influence on the engine combustion process and hence on the engine emission of pollutants. Recently simulation results by Computational Fluid Dynamics codes are used in the development and optimization of new engines by car manufacturers (automotive industry).

The in-cylinder fluid motion in internal combustion engines is one of the most important factors controlling the combustion process.

Swirl and tumble are well known approaches for in-cylinder flow enhancement. Swirl and tumble are generated in the intake stroke as a result of the inlet port shape and orientations.

Multidimensional modeling became as an important tool for investigating flow and combustion in reciprocal engines. In this type of modeling, the physical processes of flow and combustion in-cylinder governed by partial differential equations are solved with suitable boundary conditions. There are many numerical studies in the literature about multidimensional modeling of internal combustion engines.

Gosman et al. [1], numerically and experimentally, studied laminar and turbulent combustion flow in a motored engine axisymmetric reciprocating engine without combustion through a cylinder head port. Calculated and measured results were in a good agreement. They observed that the mean velocity field was influenced more strongly by the engine geometry than by the engine speed.

The investigation of the droplet and gas motion and fuel-air mixing in an axisymmetric open-chamber direct-injection engine, in the absence of combustion, were performed numerically by Gosman and Johns [2]. They found that the spray induces velocities and turbulence levels in the gas which were comparable to and sometime greater than those produced by other mechanisms such as swirl and squish.

Haworth et al. [3] developed a model by jointing probability density function and Monte-Carlo method for two- and three-dimensional turbulent non-reacting flow calculations to in-cylinder flows in reciprocating engines. The results demonstrated the feasibility of the method to complex multi-dimensional transient reciprocating turbulent flows. Influence of intake configuration on flow configuration on flow structure for two- and four-valve-per-cylinder engines were numerically investigated by using developed multi-dimensional code by Haworth et al [4].

Flow predictions in an axisymmetric inlet valve/port assembly using variants of k-ε were studied for incompressible air flow without combustion by Naser and Gosman [5]. The numerical predictions were assessed by comparing the available experimental flow field data.

Gosman and Harvey [6] analyzed fuel-air mixing and combustion in an axisymmetric direct injection diesel engine numerically. For this purpose, they improved the multidimensional model and code developed by the authors. They found that the model produces qualitatively realistic results predictions of the major phases of the combustion process, including ignition, premixed burning and diffusion burning.

Ahmedi-Befrui et al. [7] concerned with the calculation of combustion in an idealized homogeneous-charge spark-ignition engine on an axisymmetric two dimensional model. The combustion chamber was equipped with a central plug and inlet exhaust valve. Calculations were performed for a range of engine speeds, ignition timings and fuel-air ratios.

Manuscript received May 9, 2007. This work was supported by the Scientific Research Projects of Gazi University under Grant 06/2006-03. N. Dinler is with Gazi University, Faculty of Engineering and Architecture, Department of Mechanical Engineering, 06570, Maltepe, Ankara, Turkey (corresponding author to provide phone: +90.312. 231 74 00 ext. 2402 or 2422; fax: +90.312.231 98 10; e-mail: ndinler@gazi.edu.tr).

N. Yucel, is with Gazi University, Faculty of Engineering and Architecture, Department of Mechanical Engineering, 06570, Maltepe, Ankara, Turkey (e-mail: nuyucel@gazi.edu.tr).

They found that the combustion affects the mean flow turbulence structure. However, its effects on the turbulence intensity were small. In addition to that they showed that the combustion predictions were sensitive to the coefficient C_3 in the turbulence model.

Ahmadi-Befrui and Gosman [8] numerically investigated three variants of the k-ε turbulence model for the axisymmetric, disc-chamber four stroke engine model. The comparisons were made on the turbulence parameters, namely the intensity, length scale and dissipation time scale for model engine operating without combustion at 200 rpm and compression ratios of 3.5 and 6.7.

In order to simulate the mass flow rate and flow pattern of the induction system in an internal combustion engine, a three dimensional multi-dimensional code was developed by Sugiura et al. [9]. Computed velocities and static pressures obtained from simulations were in good agreement with the experimental data.

Computation of the three-dimensional flow in the intake ports of and the cylinders of the real engines, including moving valves and piston was carried out by solving the Navier-Stokes equations by Naitoh et al. [10]. No explicit turbulence models were used. The computational results were compared with experimental velocity data obtained by using Laser Doppler Velocimetry (LDV).

A combined experimental and computational study of the steady flow through a straight generic internal combustion engine inlet port was done by Chen et al [11]. Computational predictions were performed using a commercial CFD program. Comparisons of the numerical predictions with the experimental data indicated that the mean flow features were accurately predicted in many parts of the flow field.

Watkins et al. [12] investigated flow and premixed turbulent combustion in axisymmetric engine cylinders, by using a numerical model based on a modified PISO algorithm incorporating second-order bounded spatial differencing. They demonstrated that their model captures correctly the qualitative behavior of the flame near a solid wall, in marked contrast to many existing models.

By employing probability density function approach, a turbulent combustion model was developed for spark ignition engines with large variations in mixture strength by Ranasinghe and Cant [13]. The model was compared to experimental data for a homogeneous charge engine and showed good agreement.

In order to improve predictability of joint effects of shear, compression and swirl on flow and turbulence, a computational study was conducted for a valveless piston-cylinder assembly by Jakirlic et al [14]. The analyses of the mean velocity and the turbulent stresses near cylinder walls and the piston revealed a strong departure from the equilibrium law of the wall not only in the separation regions but also in the attached boundary layers. Their findings showed that the use of standard wall functions was inadequate for treating the wall boundary conditions.

Celik et al [15] made a review of computations based on large eddy simulation (LES) and concluded that this method has great potential on this kind of application.

Fluid flow calculations of the intake and compression stroke of a four-valve direct-injection Diesel engine were carried out with different combustion chambers using a commercial CFD code by Payri et al [16]. The calculated flow field with different combustion chambers was compared with laser doppler velocimetry measurements. The model results were found to be reasonable accurate.

In this study, fluid flow and combustion in a spark ignition engine were numerically investigated. Multi-dimensional engine modeling method was developed to calculation of combustion in an idealized version of axisymmetric homogeneous-charge spark ignition engine. Turbulent flow in-cylinder with a central spark plug and inlet/exhaust valve is considered. For the fluid flow, standard k-ε turbulence model and for the turbulent combustion, eddy break-up model are utilized. The effects of engine speed and valve angle on the fluid flow and combustion are analyzed for constant air/fuel ratio and compression ratio.

For the present purposes we have developed a computer code based on SIMPLE algorithm.

II. Problem Definiton and Formulation

The configuration of the cylinder with its ports is shown in Fig. 1. The cylinder is axisymmetric and inlet and exhaust valves are located at the cylinder axis. In order to obtain flow field and combustion characteristics, the ensemble-averaged differential form of continuity, momentum, enthalpy, standard k-ε and chemical reaction equations are solved with appropriate boundary conditions. Turbulent combustion is simulated by eddy break-up model.

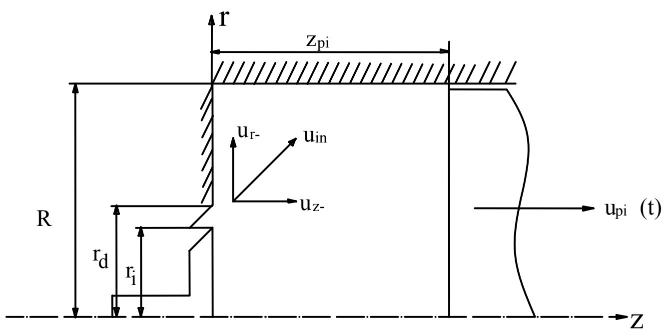


Fig. 1 Schematically description of piston-cylinder assembly of the problem

The equations were given in a form of general transport equation type. For a general ϕ dependent variable, independent variables and source terms were given in Table I [17], [18].

$$\frac{\partial(\rho\phi)}{\partial t}+\frac{\partial(\rho u_z\phi)}{\partial z}+\frac{1}{r}\frac{\partial(r\rho u_r\phi)}{\partial r}=\frac{\partial}{\partial z}(\Gamma\frac{\partial\phi}{\partial z})+\frac{1}{r}\frac{\partial}{\partial r}(r\Gamma\frac{\partial\phi}{\partial r})+S_\phi \quad (1)$$

Γ : Diffusion coefficient of ϕ variable

S_ϕ : Source term for ϕ variable

TABLE I
GENERALIZED EQUATIONS

Equation	ϕ	Γ	S_ϕ
Continuity	1	0	0
r-momentum	u_r	μ_{eff}	$-\dfrac{\partial P}{\partial r}-\mu_{eff}\dfrac{u_r}{r^2}+S_t^r$
z-momentum	u_z	μ_{eff}	$-\dfrac{\partial P}{\partial z}+S_t^z$
Turbulence kinetic energy	k	$\dfrac{\mu_{eff}}{\sigma_k}$	$G-\dfrac{2}{3}(\mu_{eff}\nabla\cdot\mathbf{V}+\rho k)\nabla\cdot\mathbf{V}-\rho\varepsilon$
Turbulence kinetic energy dissipation rate	ε	$\dfrac{\mu_{eff}}{\sigma_\varepsilon}$	$\dfrac{\varepsilon}{k}(C_1G-C_2\rho\varepsilon)$
Energy	h	$\dfrac{\mu_{eff}}{\sigma_{n,t}}$	$\dfrac{\partial P}{\partial t}$
Chemical reaction	m_f	$\dfrac{\mu_t}{\sigma_{n,t}}$	$-R_f$
Turbulence source terms	$S_t^r=\left(\begin{array}{l}\dfrac{\partial}{\partial z}(\mu_{eff}\dfrac{\partial u_z}{\partial r})+\dfrac{1}{r}\dfrac{\partial}{\partial r}(r\mu_{eff}\dfrac{\partial u_r}{\partial r})-\mu_{eff}\dfrac{u_r}{r^2}\\-\dfrac{2}{3}\dfrac{\partial}{\partial r}\left(\mu_{eff}\nabla\cdot\mathbf{V}+\rho k\right)\end{array}\right)$ $S_t^z=\left(\begin{array}{l}\dfrac{\partial}{\partial z}(\mu_{eff}\dfrac{\partial u_z}{\partial z})+\dfrac{1}{r}\dfrac{\partial}{\partial r}(r\mu_{eff}\dfrac{\partial u_r}{\partial z})\\-\dfrac{2}{3}\dfrac{\partial}{\partial z}\left(\mu_{eff}\nabla\cdot\mathbf{V}+\rho k\right)\end{array}\right)$		
Turbulence kinetic energy production	$G=\mu_t\left(\dfrac{\partial u_z}{\partial r}+\dfrac{\partial u_r}{\partial z}\right)^2+2\mu_t\left(\left(\dfrac{\partial u_z}{\partial z}\right)^2+\left(\dfrac{\partial u_r}{\partial r}\right)^2+\left(\dfrac{u_r}{r}\right)^2\right)$		
Total enthalpy	$h=C_pT+\dfrac{1}{2}(u_r^2+u_z^2)+k+\sum m_jh_j$ $\sum_{all\ j}m_j=1$		
Chemical reaction source term	$R_f=A\rho\dfrac{\varepsilon}{k}\min\left[C_Rm_f,C_R\dfrac{m_{ox}}{s},C_R'\dfrac{m_{pr}}{s+1}\right]$ $A=34.9588+(-30.2774)*(1-\exp(-0.013*N))$		
Effective viscosity $\mu_{eff}=\mu_t+\mu$ Turbulent viscosity $\mu_t=C_\mu\rho k^2/\varepsilon$ $\nabla\cdot\mathbf{V}=\dfrac{1}{r}\dfrac{\partial(ru_r)}{\partial r}+\dfrac{\partial u_z}{\partial z}$			

Turbulence model constants were chosen $C_1 = 1.44$, $C_2 = 1.92$, $C_\mu = 0.09$, $\sigma_k = 1.00$, $\sigma_\varepsilon = 1.30$.

Wall functions that are applied to near wall flow are given in detail in Versteeg and Malalasekera [18].

Ignition timing is an important parameter for spark ignition engine combustion. Ignition timing advance, (IT) is calculated by the following equation which is suggested by the authors.

$$IT = 6.83635 + 12.282(1-\exp(-0.00087N))$$

Equation of state

$$P = \rho RT$$

Boundary and initial conditions that used in order to solve the equations are given below.

At the cylinder head (z = 0) (during the intake)

$$\left.\begin{array}{l}u_z = u_{z\text{-}in} \quad u_r = u_{r\text{-}in}\\ T = T_{in}= 300\text{ K} \quad k_{in}=0{,}0001u_{z_{in}}^2\\ m_f=0.055\end{array}\right\}\text{ at the valve inlet}$$

$$\left.\begin{array}{l}\varepsilon_{in}=0.1643(k_{in}^{1,5}/0.035B)\\ u_z = 0 \quad u_r = 0\\ k=0 \quad \varepsilon=0 \quad T = 450\text{ K} \quad \dfrac{\partial m_f}{\partial z}=0\end{array}\right\}\text{ on the solid surface}$$

On the piston face (z = z_{piston})

$u_z = U_{pis}(t)$ $u_r = 0$

$k=0$ $\varepsilon=0$ $T = T_{pis}$=450K $\dfrac{\partial m_f}{\partial z}=0$

On the cylinder surface (r = R)

$u_z = 0$ $u_r = 0$

$k=0$ $\varepsilon=0$ $T = 450$ K

$$\frac{\partial m_f}{\partial r}=0$$

At the symmetry axis (r = 0)

$\dfrac{\partial u_z}{\partial r}=0$ $u_r=0$

$\dfrac{\partial k}{\partial r}=0$ $\dfrac{\partial \varepsilon}{\partial r}=0$ $\dfrac{\partial T}{\partial r}=0$ $\dfrac{\partial m_f}{\partial r}=0$

Initial conditions (t=0)

$u_z = 0$ $u_r = 0$

k = 0 $\varepsilon = 0$ T = $T_{initial}$= 300 K

m_f=0.0

Reciprocating motion of piston is modeled by using moving mesh technique. All equations were converted according to conversion equation of moving mesh.

$$\xi = \frac{z}{z_{pis}(t)}$$

III. Numerical Solution Procedure

The governing equations subject to relevant boundary conditions were solved numerically using finite-volume method. The upwind technique was employed to discretize the convective terms. A computer term has been developed by using the SIMPLE [19] algorithm. In order to obtain a solution independent of the grid distribution, grid sensitivity tests were performed by tracing the cylinder pressure against crank angle. It is found that the solution becomes almost independent with 50 uniform grids in ξ direction and 30 uniform grids in the r-direction.

IV. Results and Discussion

In this study turbulent flow and combustion in an idealized homogeneous charge spark ignition engine are analyzed numerically. It is assumed that spark plug and inlet/exhaust valves are located at the centerline of cylinder. Computations are performed for three different inlet valve angles, 30^o, 45^o and 60^o, with constant engine speed N=2400 rpm, compression ratio r=9:1 and excess air ratio λ=1.0. Cylinder radius R=0.05 m, stroke L=0.09 m and r_i= 0.04 m and r_d=0.0567 m are chosen. Methane (CH_4) is used as a fuel.

In order to observe the effects of inlet valve angle on the fluid flow, the velocity vectors are plotted for three different inlet valve angles 30^o, 45^o and 60^o at the engine speed of N=2400 rpm in Fig.2. In Figure 2, a, b and c denote the inlet valve angles α= 30^o, 45^o and 60^o. Row numbers 1, 2, 3 and 4 denote crank angle θ = 45^o, 90^o, 135^o and 180^o crank angle. As seen for crank angle θ=45^o, the velocities of air/fuel mixture, is higher than those for other valve angles considered. Therefore the strength of tumble is higher for low valve angles. Increasing the crank angle causes a secondary circulating cell between the cylinder wall and the cylinder head besides the main circulation cell. The area covered by a secondary circulating cell and its strength increases with increasing valve inlet angle. The in-cylinder turbulence is predominantly determined by the characteristics of the turbulence generated within the shear layers of intake jet. When the piston moves toward the bottom dead center, the velocity of piston decreases and reaches zero, so the mixture velocities and strength of the circulation cells decrease (Fig.2).

In Fig. 3, the fuel mass fraction contours are given at different crank angles starting from the beginning of the combustion. In the figures the contour values are represented by the level numbers. The level numbers and the values of the levels were given below the figures. In Fig. 3, a, b and c denote the inlet valve angles α= 30^o, 45^o and 60^o. Row numbers 1, 2, 3, 4, 5, 6, 7 and 8 denote crank angle θ = 343^o,

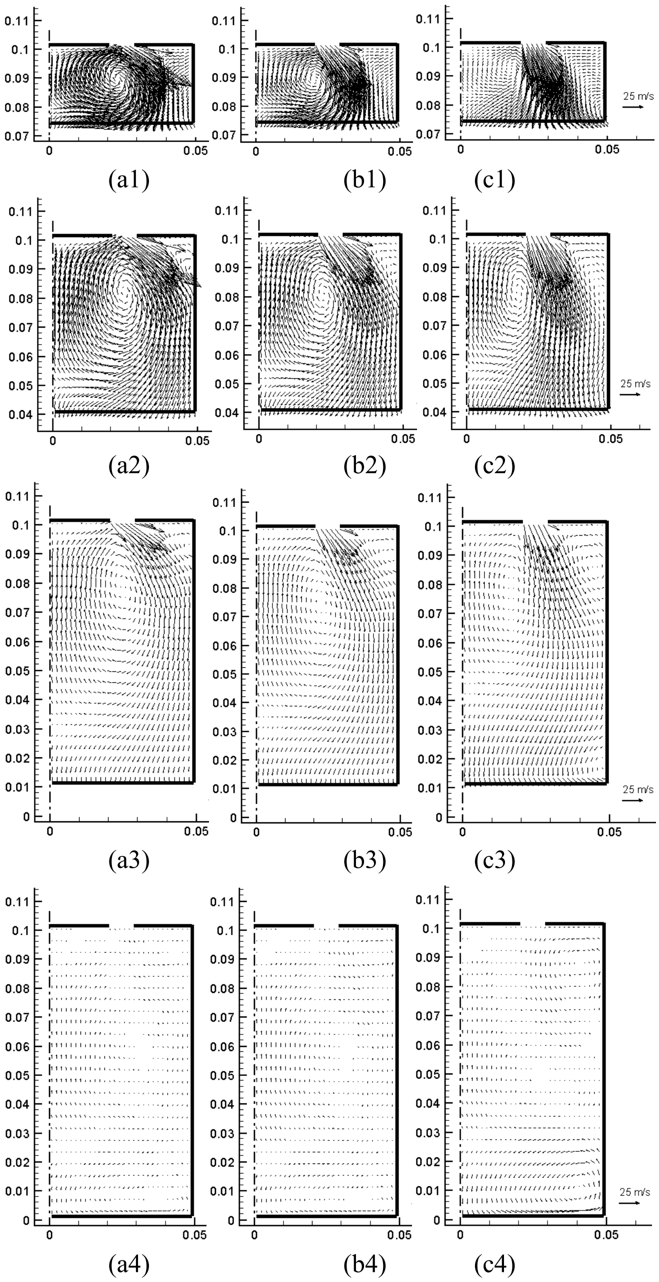
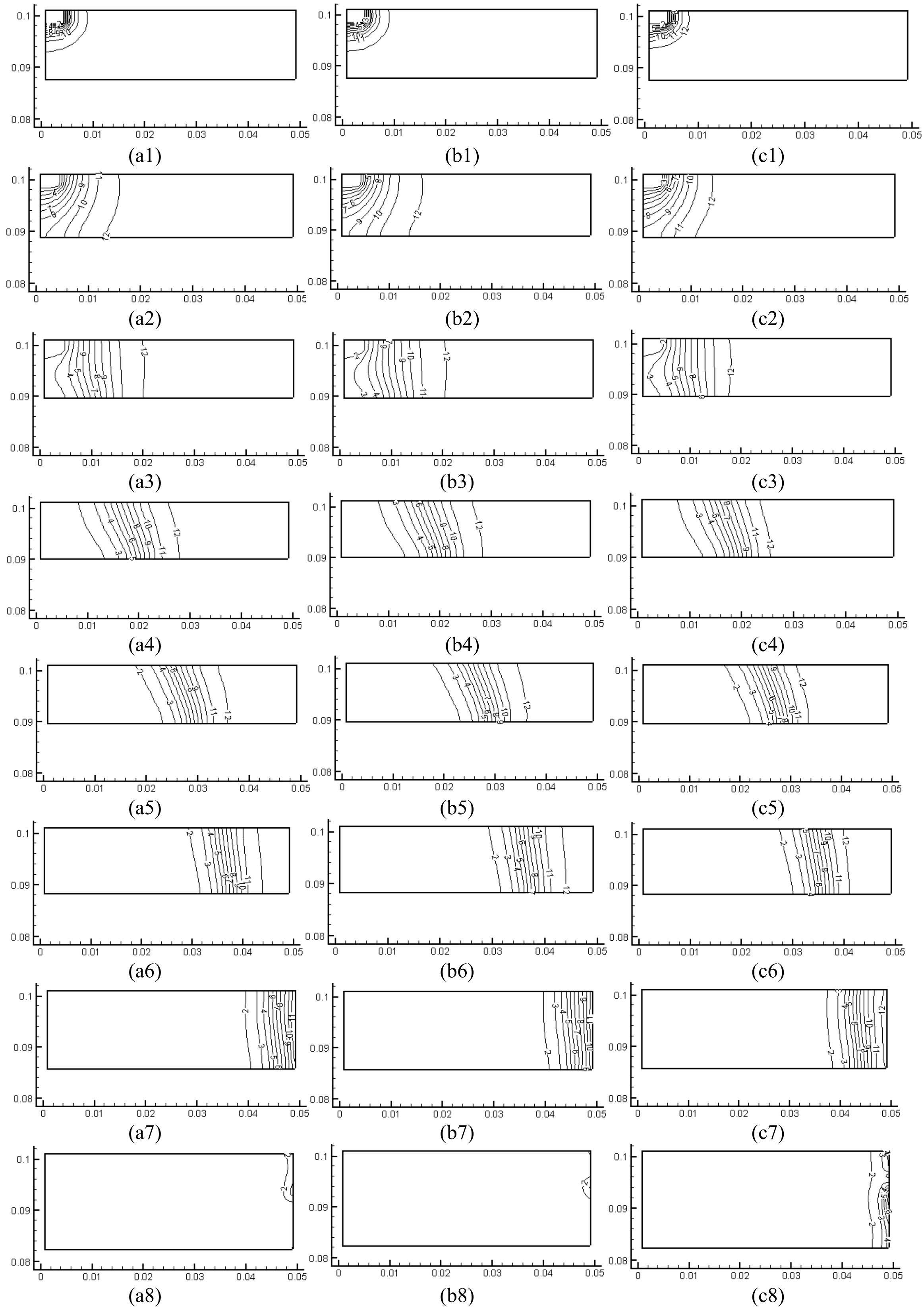


Fig. 2 Velocity vectors of flow during intake stroke for N=2400 rpm, r=9:1. (a, b and c denote the inlet valve angles α= 30^o, 45^o and 60^o. Row numbers 1, 2, 3 and 4 denote crank angle θ = 45^o, 90^o, 135^o and 180^o crank angle)

348^o, 353^o, 360^o, 367.5^o, 375^o, 384^o and 392^o crank angle. Ignition time is calculated depending on the engine speed, so the ignition started at the same crank angle (343^o) for the all cases considered. Combustion is started assuming the 70 of fuel consumed at the ignition time in the region around the spark plug. This region corresponds to the computation cells of (1 to 3 X 1 to 10) (r X ξ). The flame develops from the ignition points in a nearly hemispherical region and it reaches to the piston surface. Then the flame front propagates in the radial direction.



1	0.00000	4	0.01179	7	0.02357	10	0.03536	13	0.04714
2	0.00393	5	0.01571	8	0.02750	11	0.03929	14	0.05107
3	0.00786	6	0.01964	9	0.03143	12	0.04321	15	0.05500

Fig. 3 Contours of fuel mass fraction for combustion. (Rows a, b and c denote the inlet valve angles $\alpha = 30^o$, 45^o and 60^o. Row numbers 1, 2, 3, 4, 5, 6, 7 and 8 denote crank angle $\theta = 343^o$, 348^o, 353^o, 360^o, 367.5^o, 375^o, 384^o and 392^o crank angle)

For inlet valve angle $\alpha = 30^o$, combustion velocity is higher than that for $\alpha = 45^o$ and $\alpha = 60^o$. The combustion velocity is slowest for the valve angle $\alpha = 60^o$. However, the fuel consumption dependency was weakly related to the valve angle.

NOMENCLATURE

Symbol	Quantity
α	valve angle
ε	turbulent kinetic energy dissipation rate
ϕ	dependent variable in general transport equation
λ	excess air coefficient
θ	crank angle degree
Γ	diffusion coefficient of ϕ variable
μ	dynamic viscosity
μ_{eff}	effective viscosity
ρ	density
σ_k, σ_ε	turbulence model constants
$\sigma_{n,t}$	turbulent Prandtl number
CA	crank angle
C_1,C_2,C_μ	turbulence model constants
C_R, C'_R	model constants in combustion model
h	enthalpy
k	turbulent kinetic energy
L	stroke
m_f	mass fraction of fuel
m_{ox}	mass fraction of oxygen
m_{prod}	mass fraction of products
N	engine speed
r	compression ratio
R	cylinder radius
R_f	chemical reaction source term
s	stoichiometric oxygen requirement
S	source term
T	temperature
u_r	radial velocity component
u_z	axial velocity component

REFERENCES

[1] A. D. Gosman, A. Melling, J. H. Whitelaw, P. Watkins, “Axisymmetric flow in a motored reciprocating engine,” *Heat and Fluid Flow IMechE,* vol. 8 n.1, pp. 21-31, 1978.

[2] A. D. Gosman, R. J. R. Johns, “Computer analysis of fuel-air mixing in direct-injection engines,” *SAE Transactions*, 800091, pp. 581-596, 1980.

[3] D. C. Haworth, S. H. El Tahry, “Probability density function approach for multidimensional turbulent flow calculations with application to in-cylinder flows in reciprocating engine,” *AIAA Journal*, vol. 29, n. 2, pp. 208-218, 1991.

[4] D. C. Haworth, S. H. El Tahry, M. S. Huebler, S. Chang, “Multidimensional port-and-cylinder flow calculations for two-and four-valve-per-cylinder engines: influence of intake configuration on flow structure,” *SAE Transactions*, 900257, pp. 647-678, 1990.

[5] J. A. Naser, A. D. Gosman, “Flow prediction in an axisymetric inlet valve/port assembly using variants of k-ε,” *Proc. Instn. Mech. Engrs.,* vol. 209, pp. 57-69, 1995.

[6] A. D. Gosman, P. S. Harvey, “Computer analysis of fuel-air mixing in an axisymmetric D. I. diesel,” *SAE Transactions*, 820036, pp. 121-141, 1982.

[7] B. Ahmadi-Befrui, A. D. Gosman, F. C. Lockwood, A. P. Watkins, “Multidimensional calculation of combustion in an idealised homogeneous charge engine: a progress report,” *SAE Transactions*, 810151, pp. 636-651, 1981.

[8] B. Ahmadi-Befrui, A. D. Gosman, “Assesment of variants of the k-ε turbulence model for engine flow applications,” *International Journal for Numerical Methods in Fluids*, vol. 9, n.9, pp. 1073-1086, 1989.

[9] S. Sugiura, T. Yamada, T. Inoue, K. Morinishi, N. Satofuka, “Numerical analysis of flow in the induction system of an international combustion engine – multi-dimensional calculation using a new method of lines,” *SAE Transactions*, 900255, pp. 618-628, 1990.

[10] K. Naitoh, H. Fujii, T. Urushihara, Y. Takagi, K. Kuwahara, “Numerical simulation of the detailed flow in engine ports and cylinders,” *SAE Transactions*, 900256, pp. 629-646, 1990.

[11] A. Chen, K. C. Lee, M. Yianneskis, “Velocity characteristics of steady flow through a straight generic inlet port,” *International Journal for Numerical Methods in Fluids*, vol. 21, pp. 571-590, 1995.

[12] A. P. Watkins, S-P Li, R. S. Cant, “Premixed combustion modeling for spark-ignition engine applications,” *SAE Transactions*, 961190, 1996.

[13] J. Ranasinghe, S. Cant, “A turbulent combustion model for a stratified charged, spark ignited internal combustion engine,” *SAE Transactions*, 2000-01-0275, pp. 336-346, 2000.

[14] S. Jakirlic, C. Tropea, I. Hadzic, H. Pascal, K. Hanjalic, “Computational study of joint effects of shear, compression and swirl on flow and turbulence in a valveless piston-cylinder assembly,” *SAE Transactions*, 2001-01-1236, pp. 1402-1439, 2001.

[15] I. Celik, I. Yavuz, A. Simirnov, “Large eddy simulations of in-cylinder turbulence for international combustion engines: a review,” *Int. J. Engine Res.*, vol. 2, n. 2, pp. 119-148, 2001.

[16] F. Payri, J. Benajes, X. Margot, A. Gil, “CFD modeling of the in-cylinder flow in direct-injection diesel engines,” *Computers & Fluids*, vol. 33, pp. 995-1021, 2004.

[17] B. E. Launder, D. B. Spalding, “The numerical computation of turbulent flows,” *Comput. Meth. APPl. Mech. And Engng.*, vol. 3, pp. 269-289, 1974.

[18] H. K. Versteeg, W. Malalasekera, *An Introduction to Computational Fluid Dynamics: The Finite Volume Method*, Prentice Hall, England, 1995, pp. 1-246.

[19] S. V. Patankar, *Numerical Heat Transfer and Fluid Flow*, Hemisphere Publishing Corp., New York, USA, 1980, pp. 1-183.

Nureddin Dinler was graduated from Erciyes University Mechanical Engineering Department in 1997, in Kayseri, Turkey. He earned his M.Sc. and Ph.D. degrees in Gazi University, Institute of Science and Technology, Mechanical Engineering Department in Ankara in 2001 and 2006 respectively. Dinler is a member of Turkish Chamber of Mechanical Engineers and Society of Turkish Heat Science and Technique. The major field of study of Dinler is numerical and experimental internal combustion engines research.

Dr. Dinler has been working at Gazi University, Faculty of Engineering and Architecture, Department of Mechanical Engineering since 1998. He is the responsible research assistant of Internal Combustion Engines and Automotive Laboratory.