FLOW MODELING IN A DI DIESEL ENGINE COMBUSTION CHAMBER USING CFD

Ekrem Buyukkaya
Sakarya University
ebkaya@sakarya.edu.tr

Abstract
Turbulent flow field in a four cylinder direct injection (DI) diesel engine combustion chamber has been simulated using computational fluid dynamics (CFD). A commercial CFD code, namely Fluent was used to model 3D flow field. Three different geometrical shapes of combustion chamber were considered in the flow analysis in order to clarify the effect of combustion chamber geometry on the flow field characteristics. The simulation results showed that the bowl shapes of the combustion chambers greatly influenced the pressure, velocity and temperature distributions at the end of the compression stroke. Therefore, it has been concluded that geometry of combustion chamber could be optimized using CFD.

Keywords: Combustion chamber,Diesel piston, CFD.

1. Introduction

The in-cylinder fluid dynamics in internal combustion engines have been shown to play an important role during the combustion process [1, 2, 3]. In particular, in-cylinder fluid dynamics contribute to the fuel-air mixing which is one of the most important components for the control of the fuel burning rate for diesel engines [4]. Mixture formation mainly depends on combustion chamber configuration and distribution of fuel jet because combustion chamber configuration can control air motion within the combustion cavity (swirl and squish) and its air motion can affect distribution of the fuel jet.

The in-cylinder fluid dynamics have also been shown to significantly affect: the ignition delay, the magnitude of the premixed burn, the magnitude and timing of the diffusion burn, and the emissions of nitrous oxide and soot [5, 6]. The fluid dynamics which contribute to the combustion process in diesel engines are very complex and have been characterized with many different tools. With the advent of more powerful computers, now mathematical models accept increasingly as design and optimization tools for engine development. To accepting a mathematical tool for engine design, it is necessary to validate the model results by experimentation. Developments in engine simulation technology have made the virtual engine model a realistic proposition [7]. Today the usage of CFD codes has developed and these codes can be used to engine simulation [8, 9]. The numerical solution of in-cylinder gas flows using CFD has become very prominent in the last decade [10, 11, 12]. The most common tool for the spatial characterization of these gas flows is the use of velocity vector plots. Another popular method for spatial characterization is through two dimensional contour plots of scalar quantities such as turbulence kinetic energy and fresh air concentrations (this captures the mixing of in-cylinder gases). Since CFD generates data at every grid point in the cylinder, it is natural to volume average the in-cylinder flow field and plot it as a function of crank angle. This aids in describing the fluid motion with a single value as a function of time. This volume averaging includes scalar quantities such as mechanical and turbulence kinetic energies and the angular momentum of the fluid about all three axes (swirl and tumble ratios).

A mixture formation in an SI Engine is a complex phenomenon that is involved by large number of design and operating variables, so it is necessary to understand the in-cylinder flow characteristics for reliable designing, fuel-efficient and low emission engine. In addition, the sensitivity of the flow distribution turns back to the shape of intake port and combustion chamber so is gotten higher efficiency with better design [13, 14, 15]. In the flow within a cylinder, there are two types of motion: swirl flow commonly found in diesel engines and tumble flow commonly found in gas engines. In both cases, rotational motion occurs about an axis, though the position of the respective axis is different. In the case of swirl flow, the axis is coincident with the cylinder axis [16]. In the case of tumble, the rotation axis is perpendicular to the cylinder axis and more complex, thus making tumble flow more difficult to control than swirl flow. In order to generate swirl or tumble motion, fluid enters the combustion chamber from the intake ports. The kinetic energy associated with this motion is used to generate turbulence for mixing of fresh oxygen with evaporated fuel. The more turbulence generated lead to the better mixture of air and fuel. However, too much tumble (or swirl) can displace the flame used to ignite the fuel, cause irregular flame propagation, or result to less fuel combustion [17].

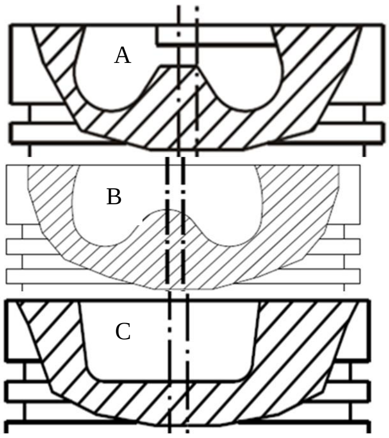
In high-speed direct-injection Diesel engines, the flow conditions inside the cylinder at the end of the compression stroke, near top dead center (TDC),
are critical for the combustion process [18]. These are determined by the air flowing into the cylinder through the intake valves during the induction process and by its evolution during the compression stroke. Several calculations of the flow in engine cylinders have been previously presented. Zur Loye et al. [19] and Kono et al. [20] performed calculations of the compression stroke. In particular, Kono et al. [20] presented an analysis of the swirl intensity effects on spray formation and obtained reasonable agreement with experimental data. However, since the intake process was not included in the calculation, the initial swirl was imposed as a parameter. It is mainly the way to estimate the initial turbulence, which differentiates the two studies. Zur Loye et al. [19] obtained it by means of a simplification of the ***kï Ů***model, while Kono et al. [20] used interpolation from experimental results. Some authors calculated the intake stroke by introducing considerable simplifications to the process. Gosman et al. [21] and Wakisaka et al. [22] considered the valve as a single moving plate but the modeled flow field was very different from the experimental results. Mao et al. [23] calculated the intake and compression strokes for an axi-symmetric case using a finite element method but did not present any validation. Arcoumanis et al. [24] calculated the compression stroke for two different piston bowls, initializing their computation at IVC of induction with a solution obtained previously with simplified flat-topped piston geometry.

The temperature and the pressure of the combustion chamber are very high, while the cyclic time is very short. Only in the past few years, did there appear some experimental methods. Ahmad et al. [25], devised a spark-discharge probe to measure the velocity and temperature fields in engines. Using the hot wire anemometer and a fast response probe, Reuss et al [26] obtained instantaneous planar measurements of velocity and large-scale vorticity and strain rate in an engine. The progresses in experimentation provided possibilities for further numerical modeling. Watkins [27] calculated the flow field and studied the heat transfer in the combustion chamber of a reciprocating engine using the SIMPLE algorithm. The top of the piston is flat and the modeling is restricted to two-dimensional laminar flow. In his further studies [28], he considered the turbulent effects and studied the flow and heat transfer in piston cylinder assemblies. In the last decade, the complex geometries are considered using more precise methods. Takenaka et al [29] used the finite element method to compute the three-dimensional in-cylinder flow with intake port in a DI diesel engine.

It is the aim of this paper to present results of an extensive CFD study of the flow characteristics inside the cylinder equipped with different piston configurations cylinder of a heavy-duty direct-injection Diesel engine. The simulation results showed that the bowl shapes of the combustion chambers greatly influenced the pressure, velocity and temperature distributions at the end of the compression stroke.

2. Combustion chamber model description

Three different types of piston bowl were modeled in the numerical simulations as shown in Fig 2. These geometries are usually considered to achieve optimal combustion conditions from the practical point of view. The first two models represent a toroidal shape, whilst the third one is simply rectangular shape of combustion chamber.



3. Numerical Solution

The main aim of this study is to numerically clarify the effect of combustion chamber on the flow field near TDC. For this purpose, CFD simulations were performed using Fluent v6.3.26, a widely used commercial code. The geometrical models and their meshes are generated with GAMBIT 2.2. Both of them are available at the Applied Fluid Mechanics Laboratory at Sakarya University. The CFD code solves numerically the unsteady Reynolds-averaged Navier-Stokes equations using an unstructured finite-volume method. The numerical algorithm used is SIMPLE, in which the pressure parameter is discretized utilizing the second-order upwind scheme. An adaptive (varying) time step was adopted for different zones of computation. For example velocity gradients and the velocities are relatively very high near discharge zone, where the time step has to be small enough so that one can capture general structure of flow field which is rapidly changing with time. However, larger time steps were used during both intake and compression strokes on order to reduce computational time. The boundary conditions for pressure and temperature were acquired from the experiments [30], and they have been assumed uniformly distributed over the sections considered. Swirling effects were not considered in the simulations. The turbulence intensity was set 10% of the mean flow, and the integral length scale was estimated with the mixing length model of Prandtl [31]. The values of these parameters describing turbulence are consistent with the models developed by Henriot et al. [32] and Inoue et al. [33]. The standard model parameters of k-ε turbulence model were used as described in references [34]. Constant pressure boundary conditions were specified at both inlets and the walls of the intake ports and the lateral walls of the valves were assumed to be adiabatic. In addition, constant temperature boundary conditions were specified for the cylinder wall, the head and the piston crown that form the walls of the combustion chamber.

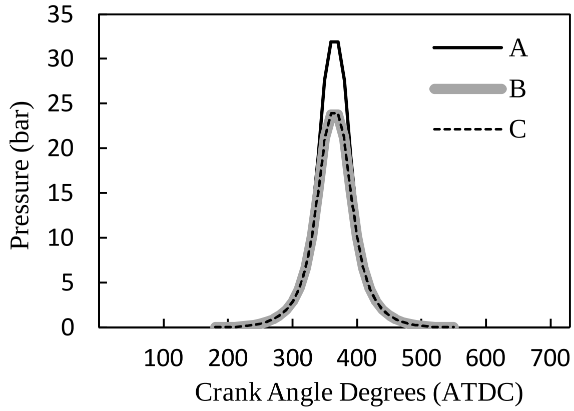
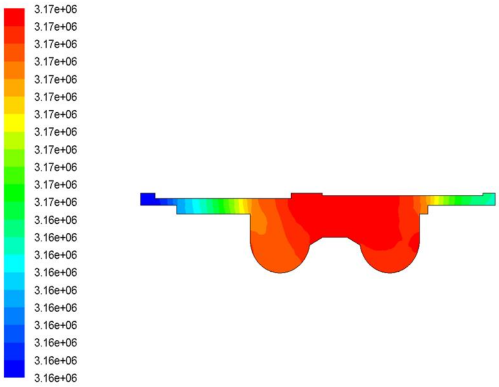


Fig: Calculated cylinder pressure versus crank angle compared for three pistons.

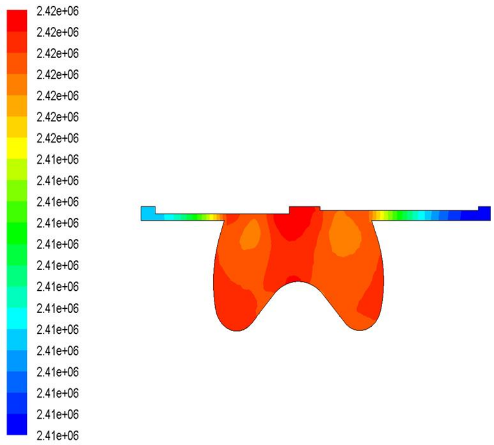
4. Results and discussion

In this study, the temperature, pressure and air flow velocities in the combustion chamber of engine are analyzed for the three piston bowl configurations described in Section 2.

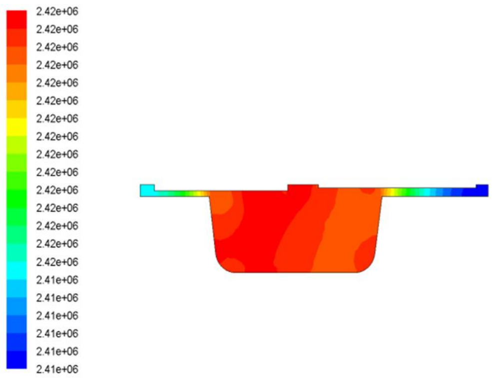
Pressure



(a)

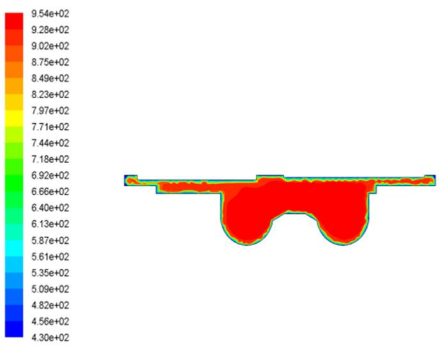


(b)

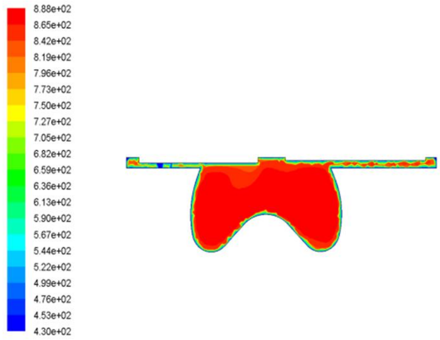


(c)

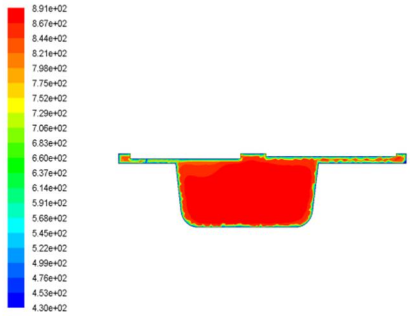
Temperature



(a)



(b)



(c)

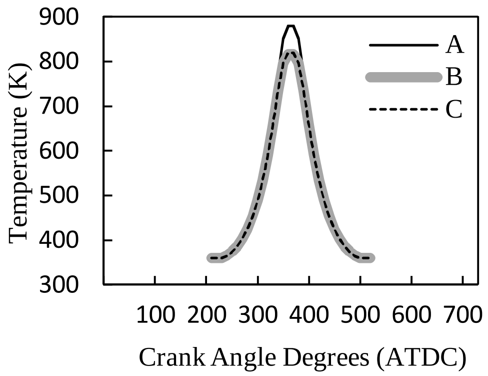
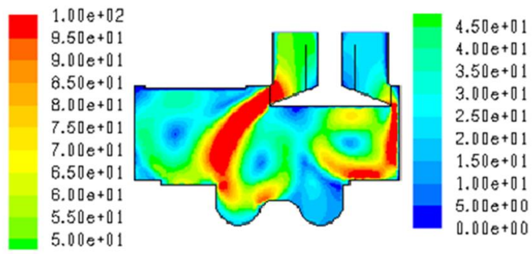
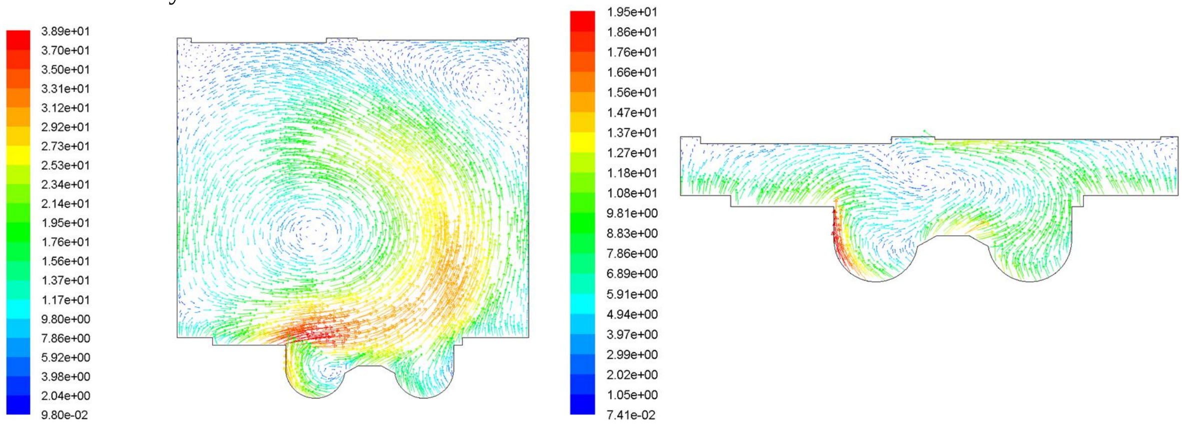


Fig: Calculated cylinder temperature versus crank angle compared for three pistons.

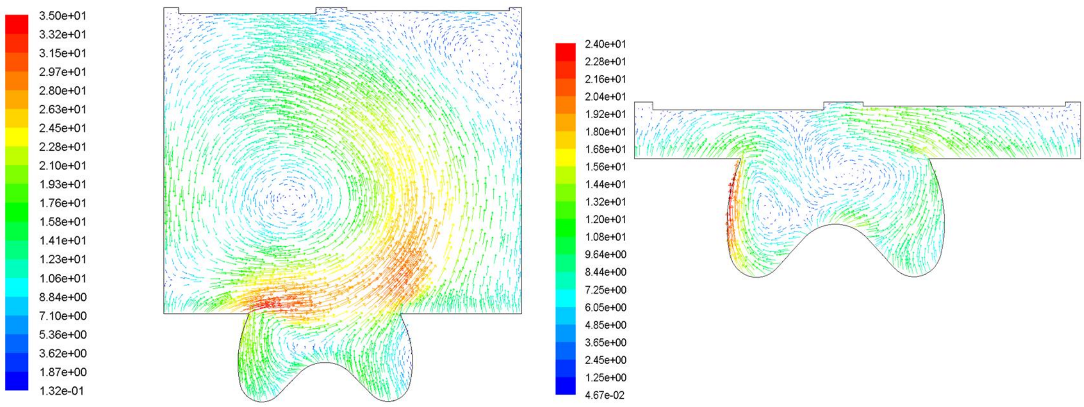


The high velocity part more than 100 m/s exist left side of the intake valve and near the wall. The very low velocity part exits at the center and have a upward velocity component.There is a strong interaction between squish and swirl near in the cases of all pistons TDC. The centrifugal forces caused by the tangential vortex impede the flow from entering radially towards the central zone of the cylinder. Hence, in the three cases considered, two toroidal vortices with opposite rotational directions appear in the combustion chamber at TDC (Fig. 5a, b, c). Belardini et al. [6] and Mao et al. [29] obtained similar results in toroidal and cylindrical combustion chambers, respectively. Arcoumanis et al. [3], however, obtained the two vortex structure only in pistons with re-entrant combustion chambers. In fact, the vortices are stronger in the case of the markedly re-entrant chamber B.

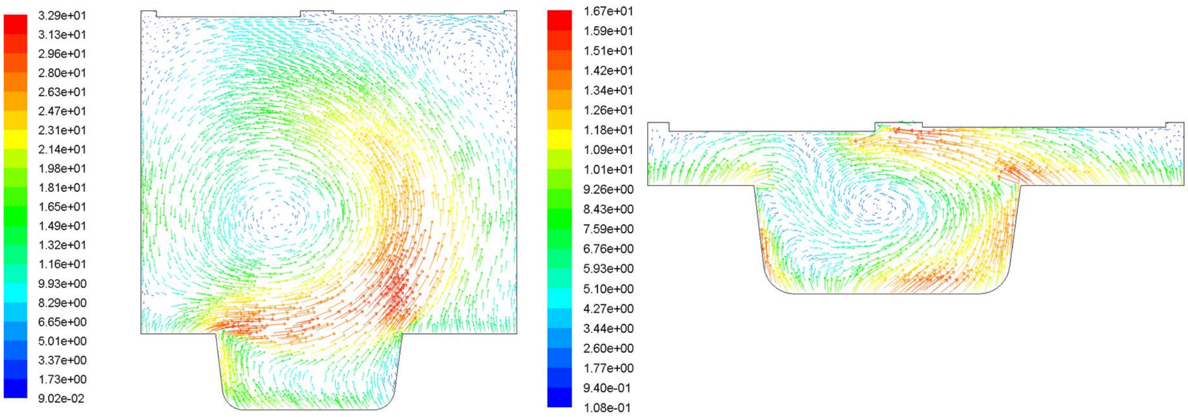
Vectorial velocity distribution



(a)



(b)



(c)

Fig.5 a, b, c: Velocity vectors at ABDC and TDC for three pistons

References

[1] Mather, D.K., "Modeling The Use of an Air-Injection for Emissions Reduction in a Direct-Injected Diesel Engine," M.S. Thesis, Department of Mechanical Engineering,
.University of Wisconsin - Madison, 1995.

[2] Monaghan, M.L., Pettifer, H.F., "Air Motion and Its Effect on Diesel Performance and Emissions," SAE 810255,1981.

[3] Shimoda, M., Shigemori, M., and Tsuruoka, S., "Effect of Combustion Chamber Configuration on In-Cylinder Air Motion and Combustion Characteristics of D.I. Diesel
Engine," SAE 850070, 1985.

[4] Godrie, P., and Zellat, M., "Simulation of Flow Field Generated by Intake Port-Valve-Cylinder Configurations - Comparison with Measurements and Applications," SAE 940521, 1994.

[5] Stephenson, P.W., Rutland, C.J., "Modeling the Effects of Intake Flow Characteristics on Diesel Engine Combustion," SAE 950282, 1995.

[6] Stephenson, P.W., Rutland, C.J., "Modeling the Effects of Valve Lift Profile on Intake Flow and Emissions Behavior in a DI Diesel Engine," SAE 952430, 1995.

[7] Li, G., S.M. Sapsford and R.E. Morgan, 2000. CFD Simulation of a DI Truck Engine Using Vectis, SAE-01-2940.

[8] Bahram, K., D.C. Haworth and Hubler, 1994. Multidimensional Port and In-Cylinder
ualizatiFlow Calculations and Flow Vison Study in an Internal Combustion Engine with Different Intake Configurations, SAE Paper, No.941871.

[9] P.Go. Drie and M. Zellat, 1994. Simulation of Flow Field Generated by Intake Port, Valve and Cylinder Configurations and Comparison with Measurements and Applications, SAE Paper, No. 940521.

[10] Arcoumanis, C., Begleris, P., Gosman, A.D., and Whitelaw, J.H., "Measurements and Calculations of the Flow in a Research Diesel Engine," SAE 861563, 1986.

[11] Gosman, A.D., "Multidimensional Modeling of Cold Flows and Turbulence in Reciprocating Engines," SAE 850344, 1985.

[12] Kuo, T.-W., "Multidimensional Port-and-Cylinder Gas Flow, Fuel Spray, and Combustion Calculations for a Port- Fuel-Injection Engine," SAE 920515, 1992.
[13] Ohm, I., H. Ahn, W. Lee, W. Kim, S. Park and D. Lee, 1993. Development of HMC Axially Stratified Lean Combustion Engine, SAE Paper No. 930879.

[14] Neusser, H.J., L. Spiegel, J. Ganser, 1995. Particle Tracking Velocimetry-A Powerful Tool to Shape the Incylinder Flow of Modern Multi-valve Engine Concepts, SAE Paper.

[15] Hicks, R.M. and G.N. Vanderplaats, 1975. Design of Low Speed Airfoil by Numerical Optimization, SAE Paper No. 750524.

[16] Das, S. and D. Chmiel, 2001. Computational and Experimental Study of In-Cylinder Flow in a Direct Injection Gasoline (DIG) Engine, 11th International Multidimensional Engine modeling Userŝ Group Meeting Agenda, Detroit, USA, March 4.

[17] Baysal, O., 1991. Multidisciplinary Application of Computational Fluid Dynamics, winter Annual Meeting Atlanta, ASME , paper No. 129.

[18] Heywood JB. Fluid motion within the cylinder of internal combustion engines. The 1986 Freeman Scholar Lecture. J Fluids Eng 1986;109(1987):3ï 35.

[19] Zur Loye AO, Siebers DL, Mckinley TL, Ng HK, Primus RJ. Cycle-resolved LDV measurements in a motored Diesel engine and comparison with kï e model predictions. SAE 890618, 1989.

[20] Kono S, Terashita T, Kudo H. Study of the swirl effects on spray formations in DI engines by 3D numerical calculations. SAE 910264, 1991.

[21] Gosman AD, Tsui YY, Watkins AP. Calculation of three dimensional air motion in model engines. SAE 840229, 1984.

[22] Wakisaka T, Shimamoto Y, Issihiki Y. Three-dimensional numerical analysis of in-cylinder flows in reciprocating engines. SAE 860464, 1986.

[23] Mao Y, Buffat M, Jeandel D. Simulation of the turbulent flow inside the combustion chamber of a reciprocating engine with a finite element method. J Fluid Eng 1994;116.

[24] Arcoumanis C, Begleris P, Gosman AD, Whitelaw JH. Measurements and calculations of the flow in a research Diesel engine. SAE 861563, 1986.

[25] T. Ahmad et al, A spark-discharge probe for velocity measurements in engines, Flows in Internal Combustion Engines, II, (1984), p. 67.

[26] D. L. Reuss et al, Instantaneous planar measurements of velocity and large-scale vorticity and strain rate in an engine using particle-image velocimetry, SAE 890616, (1989).

[27] A. P. Watkins, Flow and heat transfer in piston cylinder assemblies, Ph.D. thesis, Univeristy of London, (1977).

[28] Y. Takenaka et al, Three dimensional computation of in-cylinder flow with intake port in DI diesel engine, COMODIA 90, (1990), p. 425.

[29] G. Ryskin, L.G. Leal, Orthogonal mapping, J. Comp. Phys., 50, (1983), 71-100.

[30] Salavert JM. Estudio te_orico-experimental de la combusti_on en motores Diesel de inyecci_on directa, de 2 litros de cilindrada unitaria fuertemente sobrealimentados. Tesis Doctoral, Departamento de M_aquinas y Motores T_ermicos de la Universidad Polit_ecnica de Valencia, 1992.

[31] Launder BE, Spalding DB. Lectures in mathematical models of turbulence. Academic Press Inc; 1972.
[32] Henriot S, Le Coz JF, Pinchon P. Three dimensional modelling of flow and turbulence in a four-valve spark ignition engineï ï Comparison with LDV measurements. SAE 890843, 1989.

[33] Inoue S, Kobayasi K, Akatsuka F, Fukumori E. Calculation of the in-cylinder flow and heat transfer in DI and IDI Diesel engines. SAE 890667, 1989.

[34] Engin, T., "Study of Tip Clearance Effects in Centrifugal Fans with Unshrouded Impellers Using Computational Fluid Dynamics", Proc. ImechE. Part A: Journal of Power and Energy, 220(6), 599-610 **(2006)**.