Altin Dorri, Andonaq Lamani (Londo), Artan Hoxha

ISSN 0350-350X
GOMABN 47, 3, 351-371
Izvorni znanstveni rad/Original scientific paper
UDK. 532.528 : 621.436.038.5-225.8 : 518.5

INFLUENCE OF HOLE GEOMETRY IN THE CAVITATION PHENOMENA OF DIESEL INJECTORS, A NUMERICAL INVESTIGATION

Abstract

In order to evidenciate the effects of the hole shape in the generation of the spray, in high pressure diesel nozzles, a numerical investigation has been carried out. In particular the effect of cylindrical and non-cylindrical hole geometry on spray characteristics has been investigated in order to understand the relationship between nozzles geometry and in-nozzle flow features. Different hole shapes (convergent conical, divergent conical and cylindrical) are tested using the finite-volume CFD code FLUENT 5.5 at different fuel injection pressures. Simulation was done with pressure between 40 MPa and 140 MPa, and 50 µm needle lift. In the CFD simulations the pressure contours, the flux line, velocity vectors and the mass flow rate of the fuel inside the nozzle have been studied. Results have shown that variation of pressure inside of the holes changes notably according to the conicity. Volumetric fraction behavior of three injectors was very different

1. Introduction

Actual development of compression ignition engine is mainly related with the rising of injection pressure and the possibility of injecting several jets during a single injection cycle. Both modifications influence positively the engine characteristics and the emissions formation process. As a consequence the capillary diffusion in the injection systems of high pressure for Diesel Engine with direct injection has stimulated in the last decade the research in the field of complexes fluid dynamics phenomenons located inside the injectors. Especially, phenomenon related to the cavitation inside the injectors [1, 2] have assumed particular importance as it regards the generation of the spray, a crucial phenomenon to get an efficient combustion and low emission of NOx and soot. Normally cavitation is well known as a harmful phenomenon, which favors the erosion of the mechanical parts, as for example in narrow zones at the upper part of injector. But inside the nozzles (holes), it favors the pulverization of the fuel and as a consequence increases the performances regarding emissions, resulting therefore a phenomenon to be favored

[3]. The cavitation is strongly related to high injection pressures in the new injection systems (more then 100 MPa) and with the injector geometry. To get a spray constituted more and more by small bubbles goes toward geometry of multi-hole injector (also up to eleven holes) with diameters extremely reduced (order of 100 µm). The reduced sections of oil-fuel passage, elevated pressures of injection jointly with elevated angles of inclination of the holes are the causes for the formation of cavitation bubbles inside the holes of the injector [4]. Cavitation is a physical phenomenon that generate whereas the pressure of the liquid phase goes down below the vapor pressure. The formation of vapor's bubbles that follow the cavitation can be of heterogeneous or homogeneous type. If the nucleation of bubbles is favorized by the presence of micro-particles dispersed in the primary fluid or caused by the wall roughness of the duct that contains the fluid it produces heterogeneous cavitation, otherwise if it is entirely due to some conditions of nucleation it demonstrates homogeneous cavitation. The ways with which the conditions of cavitation can be created inside the high pressure injectors are so many and not all known

In order to predict and control fuel sprays related to cavitation various experimental and theoretical research have been carried out. Experimental observation [5, 6] evidenced interesting differences in the permeability of nozzle geometries and a clear resistance of the conical nozzle to cavitation. Hountalas et al. [7] setup an experimental investigation using three different nozzle hole types: a standard, a convergent and a divergent one to discern the effect of nozzle hole conical shape on engine performance and emissions. According to the experimental findings, an increase of soot and decrease of NO_x was observed for the divergent nozzle hole compared to the other two.

On the other hand due to difficulty do set up experiments related this to small size of injector holes many numerical researches are conducted for this topic and many cavitation models have been developed. Numerical researches [8, 9] are mainly focused in the individualization of the appropriate physical models both as it regards the nucleation of bubbles, also as it regards their growth and collapse. Whatever, considering physics of two-phase phenomena extremely complex and coupled to the phenomena of mass transfer among the two phases, as well as strongly correlated with the turbulence phenomena inside the fluid, unification of phenomena and equations that govern this kind of phenomena results extremely arduous. Important factor in cavitation phenomena is the *cavitation number*, defined as:

$$CN = \frac{P_{inj} - P_{out}}{P_{out} - P_{vap}} \tag{1}$$

as well as the inside geometry of the injector determines the conditions to increase the vapor bubbles generation according to the fluid dynamics condition produced inside the flow.

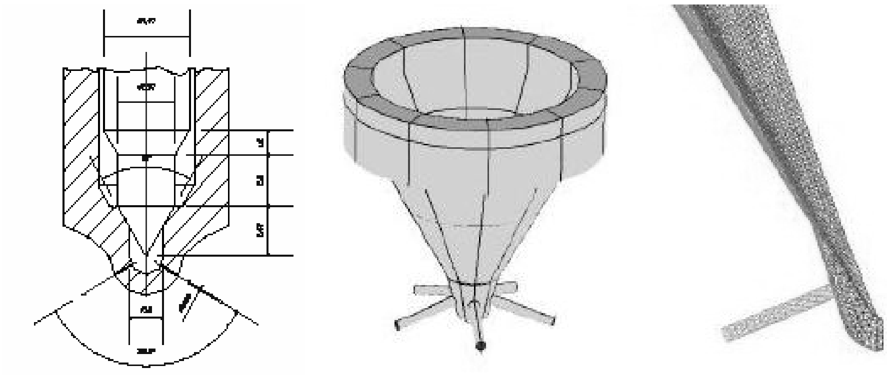
In the present work we are mainly focused to the influence of holes geometry in the cavitation phenomena. For this purpose, three holes with different conicity (cylindrical, convergent and divergent) have been investigated using the commercial CFD code *Fluent* 5.5.14 (witch is available near our Department). Results were compared in appearance with experimental work [5] and other previous works. This paper is a synthesis of [10].

2. Injector under investigation

One of the main geometric characteristics of the injector holes is the variation of the cross-section along the length, which represents the orifice conicity. The injector object of simulations are mini-sac type with five holes, as in Figure 1, with nominal diameter of the holes D = 68 µm. The examined injectors differ only from holes conicity, expressed by:

$$C = \frac{D_{in} - D_{out}}{10} \tag{2}$$

where the dimensions of inlet diameter (D_{in}) and outlet diameter (D_{out}) are express in mm. The different values examined for the coefficient of conicity C have been: C = 0 (cylindrical hole), C=+1.5 (convergent conical hole), C= -1.5 (divergent conical hole).



a) b) c)

Figure 1: Injector under study: a) Cross section of injector, b) 3D view of nozzle domain, c) 3D mesh of nozzle sector object of simulation

Based on the geometrical symmetry also on the similarity behaviors of five holes during the oil-fuel injection confirmed by [5] was useful to limit the CFD study only for a fifth of nozzles so a 72 degree sector was simulated (Fig. 1.c). This has leaded to decrease of time consumed and less CPU used for the calculation. Another reason for this simplification was the limited capability of the computer used for the simulation.

3. Cavitation model implemented in Fluent

CFD software used for the numerical simulations is *Fluent* 5.5, that it implements a model able to predict the beginning of fluids cavitation. Such cavitation model is able to model the behavior of the two phases, liquid and vapor, by solving a single set of momentum equations and the continuity equation for the secondary fluid [11]. The model implemented can predict the inception of cavitation but not the collapsing of the bubbles. The cavitation model relies on the volume fraction concept (VOF) but differs from the VOF model in two substantial aspects: i) unlike of VOF (that model two un-mixed fluids), does not assume that there is an interface between two immiscible fluids; it allows the fluids to be interpenetrating. The volume fractions of two phases (α_v and α_l) for a control volume can therefore be equal to any value between 0 and 1, depending on the space occupied by each phases. ii) The cavitation model allows mass to be transferred from one phase to another. These allow us to model the formation of vapor from a liquid. The current implementation of the cavitation model assumes a two-phase homogeneous model (no slip between the phases). Like the VOF model, it solves a single momentum equation for all phases and a volume fraction equation for the secondary phase. The volume fraction of the primary phase will be computed by subtracting the secondary-phase volume fraction from 1.

The continuity equation for the vapor phase is the basis of the volume fraction equation:

$$\frac{\partial}{\partial t}(\alpha_v)+\frac{\partial}{\partial x_i}(\alpha_v u_j)=\frac{1}{\rho_v}(\dot{m}_{vl}-\frac{d\rho}{dt}) \tag{3}$$

The liquid-phase volume fraction will be computed based on the following expression:

$$\alpha_v+\alpha_l=1 \tag{4}$$

Cavitation model implemented in Fluent hypothesizes that the phenomenon is isothermal, neglecting thermal exchanges and particularly the latent heat of vaporization, since cavitation bubbles will form in a liquid at low temperatures. Under this hypothesis the pressure within the bubble remains nearly constant at such conditions, and the change in bubble radius can be approximated by a simplified Rayleigh equation:

$$\frac{dR}{dt}=\sqrt{\frac{2(p_v-p)}{3\rho_l}} \tag{5}$$

Therefore total mass of vapor phase can be calculated as follow:

$$m_v = \rho_v \frac{4}{3}\pi R^3 n \tag{6}$$

4. Results of CFD simulations and Discussion

As we cited above only a 72 degree sector of nozzle was considered. The calculations were performed on two different grids (Figure 2), to investigate the possibility of grid sensitivity. Both 3D meshes are made with the same cell type tetragonal, but with different density of cells. The densest grid has been generated with 216440 cells, with a mean dimension of the side 11.3 µm, while the less dense grid has been generated with a number of cells equal to 30920 with a mean dimension of the side 22.6 µm. As a determinant parameter, volume fraction of vapor phase was considered for comparison (Figure 2). Since the result for both grids were almost similar was judged to use the less dense grid.

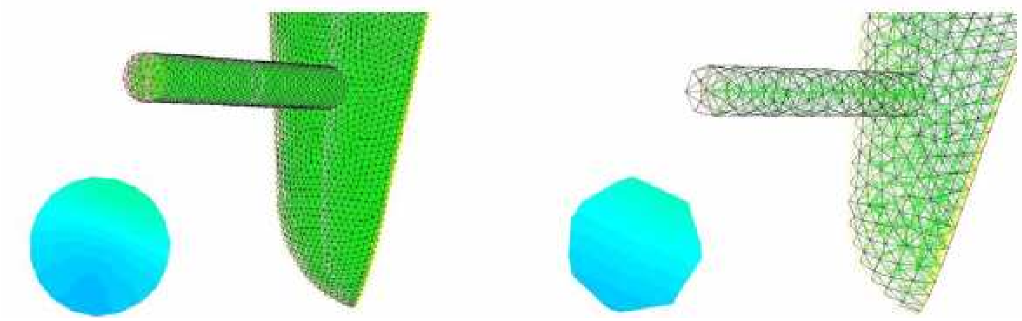


Figure 2: Mesh sensitivity for vapor volume fraction in the central cross section of hole

The inlet and outlet are modeled as pressure boundaries with 40 MPa and 140 MPa of gauge pressure at the inlet and 0,1 MPa at the outlet, while needle lift is considered to be 50 µm. The working fluid is a mixture of liquid diesel fuel and diesel vapor. All properties for both phases, including the vapor pressure, are specified. As well as the cavitation model also standard *k-ε* model is activated for the turbulent flow conditions. CFD simulations have been performed in unsteady regime, as required by the cavitation model implemented in Fluent, and the results are refered to the time of 3.5 $*10^{-3}$ sec. Simulations have pointed out start of cavitation inside the hole for all three different configurations and for both two injection pressures. In Figure 3, the distribution of static pressure on the mid-plane of the injector's sector under investigation is shown. After pass through the narrow section where an instantaneous decrease occures, from 140 MPa to 110 MPa, the pressure increases, re-stabilizing inside the mini-sac volume to have another decrease inside the exit hole. While the pressure distribution inside of the injector body has a similar behavior for all three investigated geometries, pressure distribution inside of the holes varies notably according to the conicity. As we can see from Figure 4, the pressure inside the cylindrical hole (C = 0) decrease with a gradient almost constant from hole's inlet up to the outlet section. In Figure 5 the pressure distribution inside the convergent conic hole is shown, (C = +1.5), in which is evident a pressure degrease of diesel fuel inside the hole but more gradual compared to cylindrical hole. In the case of divergent conic hole (C = -1.5) it has an instantaneous decrease of pressure from the hole's inlet, and then remain quasi-constant up to the outlet (see Figure 6).

In the Figures 7, 8, 9 are relatively represented contours of diesel vapor volume fraction inside the injectors. The behavior of the three injectors is very different. In case of cylindrical hole geometry (Figure 7) cavitation begin corresponding to the inlet corner, where, due to the instantaneous change of the fluid flow direction, by reason of a slightly rounded inlet, it has a separation of the fluid stream from the upper wall and as a consequence a pressure decrease at or below the vaporization pressure so cavitation occurs and propagates up to the outlet. This type of cavitation was pointed out also in previous jobs [1, 2, 8, 12] and is strongly related to the inlet section geometry of the hole, which influences in determinant manner the stream lines of diesel fuel in determinant manner. Flow behavior changes notably due to the hole conicity.

Comparing figures 7, 8, 9 it is evident that cavitation in the convergent hole (figure 8) is exclusively located in the outlet zone. In fact because of the hole convergence the flow lines are compacted and possibilities that there is a separation from the wall are reduced to the minimum. Although also in this case the inlet corner is rather edgy, the separation of the fluid stream from the upper wall doesn't take place and the pressure decrease inside the hole is less emphatic. Contrary phenomenon happens for the hole with divergent conicity (Figure 9). In fact, divergent geometry of the hole amplifies the tendency of flow lines to separate and therefore the cavitation occurs on the whole length of the hole, producing a notable increase of vapor fraction compared to the cylindrical hole case. In their experimental work De Risi et al. [6] observed that the main difference between three types of holes was in the jet angle of the cone spray for the first instant of the needle lift, and this is attribute to differences of cavitation over three holes, so the same that we evidence above in the numerical observation.

Simulations have been performed also regarding the injected quantity of fuel (mass flow rate) from all three typologies of holes. Cylindrical hole produces a greater flow rate compared to convergent hole since convergent geometry chokes the flow lines at the outlet. Comparing cylindrical and divergent conical hole is the cavitation phenomena predominant, that in divergent case is ten times greater compared to cylindrical hole and as a consequence this reduce the fuel flow rate at the outlet. Simulations with P_{inj} = 40 MPa have produced similar results for both as it regards contours of pressure inside various typologies of hole and initiation of cavitation. Besides standard k-ε turbulence model also Reynolds Stress Model was used for simulations that, however, has not produced appreciable different results from k-ε

model expect an increase in calculation times. The same happened with the reduplication of mesh using k-ε model, results were almost the same.

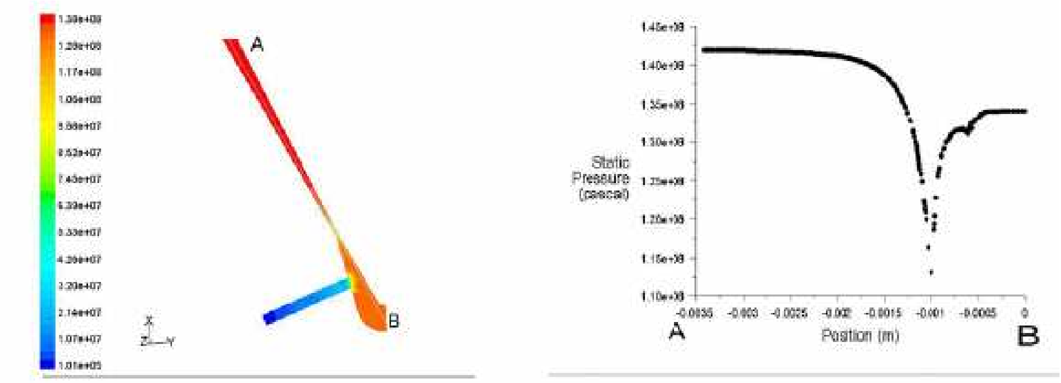


Figure 3: Contours of pressure inside the injector

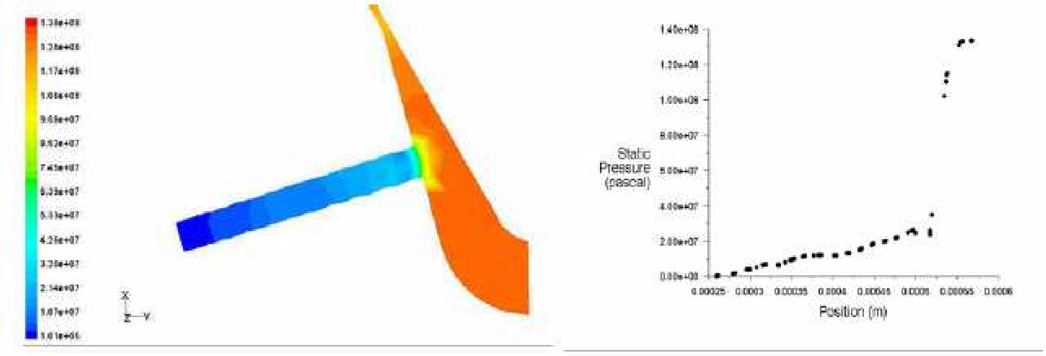


Figure.4: Contours of pressure inside the cylindrical hole (C=0)

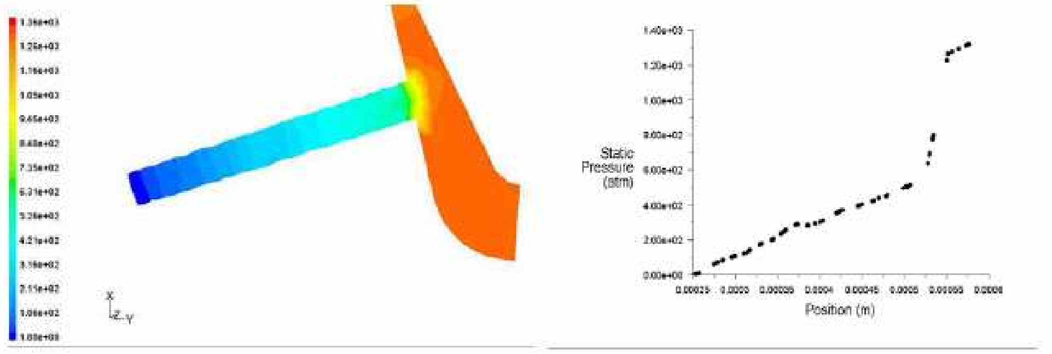


Figure 5: Contours of pressure inside the convergent conic hole (C=+1.5)

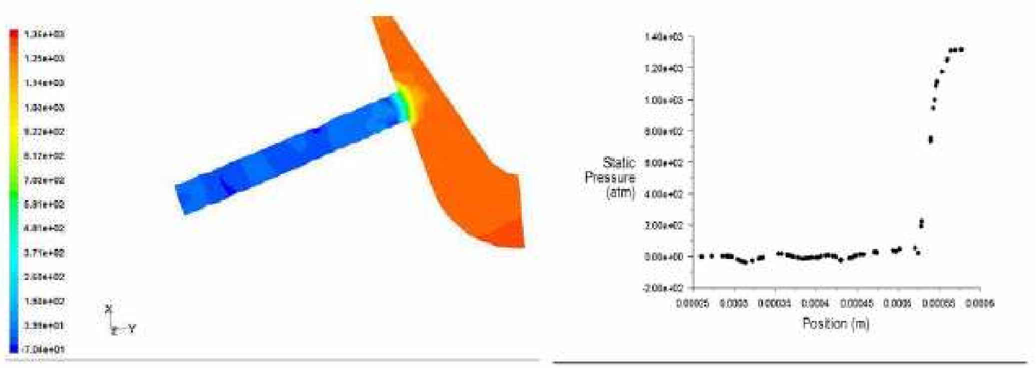


Figure.6: Contours of pressure inside the divergent conic hole (C=-1.5)

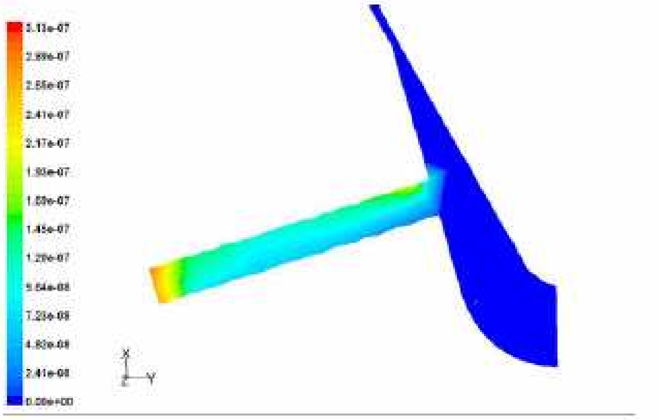


Figure.7: Contours of diesel vapor volume fraction, inside the cylindrical hole (C=0)

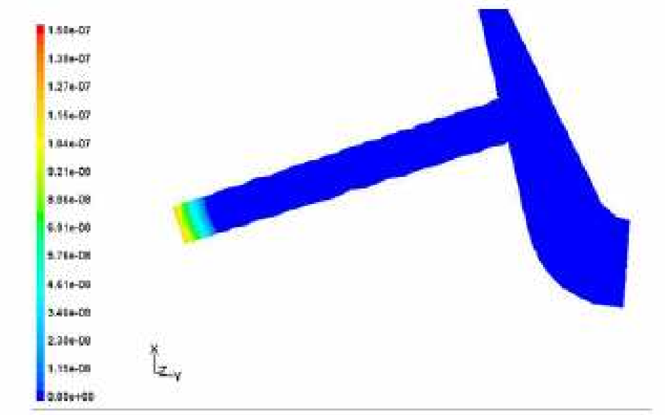


Figure 8: Contours of diesel vapor volume fraction, inside the convergent conic hole (C=+1.5)

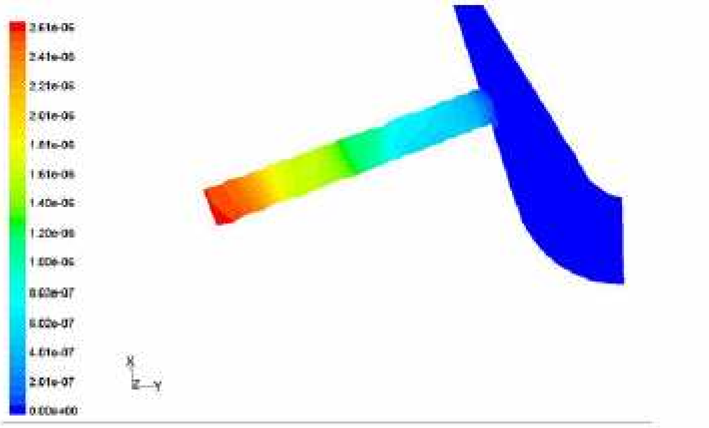


Figure 9: Contours of diesel vapor volume fraction, inside the divergent conic hole (C=-1.5)

Conclusion

The aim of this paper has been to study the influence of hole geometry in cavitation phenomena and vaporization. For this purpose three 5-hole mini-sac nozzles, different for the hole conicity, have been simulated. Numerical simulation show up increasingly characteristics and differences of cavitation flow produced by axial-symmetric mini-sac injectors of five holes with different conicity, but also limits of tools used for this numerical study. CFD simulations have put in evidence of differences regarding initiation of cavitation in all three types of holes, underlining great and substantial differences in the phenomena that cause the cavitation for considered geometries. Results of the pressure field indicate regions where cavitation is likely: regions where recirculation occurs and the pressure is near in value to the vapor pressure. Cavitation model was able to predict the inception of cavitation and differences of these phenomena in all of three holes under investigation.

Weak point of the investigation is that only 1/5 of the device was considered for simulations. Also it is necessary to underline limits of cavitation model implemented by Fluent that it doesn't allow the evolution modeling of vapor bubbles forming, not succeeding in simulating the collapse. Cavitation model of *Fluent* is also limited by not considering the thermal changes and therefore variations of temperature. The implementation of a deformable grid in time would be desirable so to be able to simulate the evolution of the flow under real operation conditions and with a moving needle inside the injector, so to be able to simulate the behavior of the flow under turbulent conditions produced by the needle movement. The possible simulations with the actual version of Fluent allow only characterizing the phenomena with a fixed needle lift positions and therefore in static conditions and not dynamic, as in real conditions that the injector operates.

However numerical modeling of cavitation is an economical method of evaluating various injector designs through a range of operating conditions, since actual test measurements for these devices are difficult to conduct.

Nomenclature

CN	-	Cavitation number
P_{inj}	Pa	Pressure of injection
P_{out}	Pa	Outlet pressure
P_{vap}	Pa	Pressure of vapor
C	-	Conicity
D_{in}	mm	Inlet diameter of the hole
D_{out}	mm	Outlet diameter of the hole
α_v	-	Volumetric fraction of the vapor phase
α_l	-	Volumetric fraction of the liquid phase
ρ_l	kg/m^3	Density of the liquid phase
ρ_v	kg/m^3	Density of the vapor phase
R	mm	Radius of the vapor bubbles
n	-	Number of bubbles per unit volume
$\dot{m}_{vl}$	kg/s	Mass transferred between liquid phase and vapor phase

References

[1] **Arcoumanis C., Flora H., Gavaises M., Kampanis N., Horroks R.**, “Investigation of Cavitation in a Vertical Multi-Hole Injector”, *SAE paper* 1999-01-0524, 1999.

[2] **Arcoumanis C., Badami M., Flora H., Gavaises M.**, “Cavitation in Real-Size Multi-Hole Diesel Injector Nozzles”, *SAE paper* 2000-01-1249, 2000.

[3] **Arcoumanis C., Whitelaw J. H.**, “Is Cavitation Important in Diesel Engine Injectors?”, Proceeding of Thiesel ‘2000, *Conference on Thermofluiddynamic processes in Diesel Engines*, Valencia 13-14 September 2000.

[4] **Kato M., Kano H., Date K., Oya T., Niizuma K**, “Flow Analysis in Nozzle Hole in Consideration of Cavitation”, *SAE paper* 970052, 1997.

[5] **De Risi A., Colangelo G., Laforgia D.**, “An Experimental Study of High Pressure Nozzles in Consideration of Hole-to-Hole Spray Abnormalities”, *SAE paper* 2000-01-1250, 2000.

[6] **Benajes J., Pastor J. V., Payri R., Plazas A. H.**, “Analysis of the Influence of Diesel Nozzle Geometry in the Injection Rate Characteristic”, *Journal of Fluids Engineering* 126, pp. 63-71, 2004

[7] **Hountalas D. T., Zannis T. C., Mavroooulos G. C., Schwarz V., Benajes J., Gonzalez C. A.,** "Use of a multi-zone combustion model to interpret the effect of injector nozzle hole geometry on HD DI diesel engine performance and pollutant emissions“, *SAE paper* 2005-01-0367, 2005

[8] **Schmidt D. P., Rutland C. J., Corradini M.**, “A Numerical Study of Cavitating Flow Through Various Nozzle Shapes”,.

[9] **Soteriou A., Andrews R., Smith M.**, “Direct Injection Diesel Sprays and the Effects of Cavitation and Hydraulic Flip on Atomization”, *SAE paper* 950080, 1995.

[10] **Dorri A.,** “Fuel injection system in Diesel engine and cavitation problems”, Master Thesis, Polytechnic University of Tirana, Tirana 2006

[11] **Fluent. Inc**. Fluent 5.5 User's Guide, *Fluent Inc*., Lebanon NH, 2000.

[12] **Qin J., Yu S. T. J., Zhang Z.C., Lai M. C.**, “Transient Cavitating Flow Simulations Inside a 2-D VCO Nozzle Using the Space-Time CE/SE Method”, *SAE paper* 2001-01-1983, 2001.

UDK	ključne riječi	key words
532.528	kavitacija strujanjem	flow cavitation
621.436.038.5	uređaji za ubrizgavanje goriva dizelovog motora	diesel engine fuel injection devices
621.436-225.8	sapnica za ubrizgavanje, oblikovanje i konstrukcija	injection nozzle, design and construction
518.5	računalne metode i modeliranje	computer methods and modeling

Authors

Altin DORRI*, MSc. Ing., e-mail: altindorri@yahoo.it; Andonaq LAMANI (LONDO), Prof. Dr., Artan HOXHA, MSc. Ing.

Polytechnic University of Tirana, Albania

Received

23.2.2009.

Accepted

24.3.2009.