ILASS 2008
Sep. 8-10, 2008, Como Lake, Italy

Paper ID ILASS08-P-10

A NUMERICAL STUDY OF SPRAY CHARACTERISTCS THROUGH VARIOUS NOZZLE ASPECT RATIO

Dong-Sik Han*, Byoung-Hwa Lee**, Young-June Chang***, Ju-Hun Song****, Chung-Hwan Jeon°

* Pusan National University, Republic of Korea, +82-51-510-3035, +82-51-582-9818, aenioes@naver.com
** Pusan National University, Republic of Korea, +82-51-510-3035, +82-51-582-9818, bhlee@pusan.ac.kr
***Pusan National University, Republic of Korea, +82-51-510-2332, +82-51-582-9818, changyj@pusan.ac.kr
****Pusan National University, Republic of Korea, +82-51-510-2332, +82-51-582-9818, jxs704@pusan.ac.kr
° Pusan National University, Republic of Korea, +82-51-510-7324, +82-51-582-9818, chjeon@pusan.ac.kr

ABSTRACT

Spray formation mechanism was controlled by a cavitation inside an injection nozzle. Nozzle geometry affects spray characteristics and atomization behaviour, which could determine engine performance and pollutant formation. A study was carried out on the influence of aspect ratio on cavitation inside a nozzle and spray behaviour in a combustion chamber. The caviation model available in Star-CD code was used to obtain cavitation behavior inside a nozzle, which was compared with previous experimental results. In this paper, a CFD approach combining multiphase Volume–of–Fluid (VOF) and k-ω model was applied. The results agree well with the experimental results indicating the potential of cavitation for enhancing of atomization and spray quality, which could increase the spray angle for homogenous mixing.

INTRODUCTION

The injector nozzle is one of the most important parts of a diesel engine. Nozzle geometry affects spray characteristics and atomization behaviour, which is decisive of engine performance and pollutant formation. The entire breakup analysis is based on aerodynamic interactions between the liquid and gas phases, and modified initial conditions that may be caused by different nozzle design can only be included by adjusting empirical constants to experimentally obtained data. However, comprehensive studies on this subject show that effects of the inner nozzle flow such as liquid phase turbulence and cavitation do have an increasing influence on primary spray breakup for modern high pressure diesel engine.

The primary breakup of liquid jets at the nozzle exit can be caused by a combination of three mechanisms: turbulence within the liquid phase, implosion of cavitation bubbles and aerodynamic forces acting on the liquid jet.

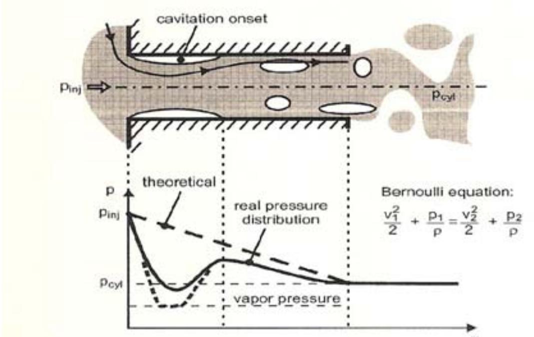

Fig. 1 Schematic illustration of cavitation formation inside the nozzle hole [1]

Due to the pressure drop across the injection nozzle, the liquid fuel is accelerated within small nozzle holes. Thereby a high level of turbulence is generated within the liquid phase that has a destabilizing effect on the jet once it exits the nozzle hole. Additionally, at sharp edges along the flow path inside the nozzle, e.g. at the inlet of the nozzle hole, the streamlines are contracted such that the effective cross-section the flow is reduced and its velocity is accelerated even more. According to Bernoulli`s law, this causes a reduction in the static pressure, and locally the static pressure may be decreased to a value as low as the vapour pressure of the fuel. This phenomenon is schematically shown in Fig. 1, where the theoretical pressure distribution along a streamline is plotted. The effect is magnified by cavitation bubbles are generated inside the injection nozzle [1].

A schematic illustration of the two-zone primary breakup model is depicted in Fig.2. A primary ligament containing both the intact liquid and the cavitation zone is injected into the combustion chamber.

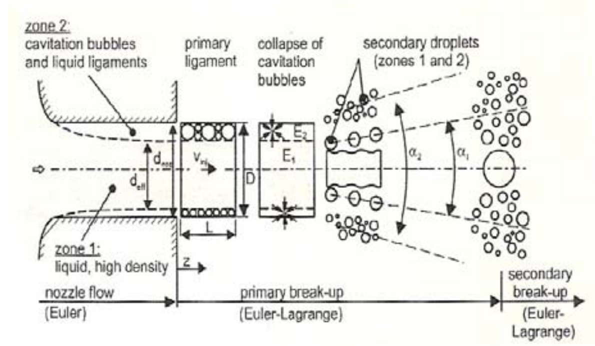

Fig. 2 Schematic illustration of the two-zone primary breakup model in the combustion chamber [2]

Due to the rise in static pressure the cavitation bubbles in zone 2 will implode such that energy is released and pressure waves are initiated that propagate both to the inner and outer surfaces of zone2. It is now assumed that the fraction of energy that reaches the outer surface, results in breakup of zone2, whereas the remaining energy fraction reaching the interface between zones 1 and 2 increases the turbulence level in the liquid zone 1 and subsequently causes breakup of zone 1. The distribution of the total cavitation energy on the two zones is assumed to be proportional to the areas of the inner and outer interfaces of zone 2.

In both zones 1 and 2, the breakup of the primary ligament into secondary droplets occurs once the collapse time of the cavitation bubbles has been exceeded. The breakup is due to the sum of turbulent kinetic energy and the energy induced by the collapse of the cavitation bubbles. The total amount of energy will be transferred into a combination of surface energy of secondary droplets and a velocity component that is perpendicular to the spray axis. The amount of surface energy controls the resulting droplet radius, and the radial velocity component is responsible for the visible spray angle [2].
Spray measurements outside the nozzle have shown that cavitation inside the nozzle causes as substantial change of relevant spray characteristics such as spray angle [3].

Because of the difficult in performing experiments for this small scale flow, time accurate solutions of flows through injector nozzles are of great interest. Numerical simulation provides as efficient method to further understand this small scale flow. In this study, we will first briefly describe our numerical method for the simulation of cavitationg flows, and then we used the method to predict the dynamic performance of cavitating flows in injection nozzles as aspect ratio varies. The present numerical approach to the solution of cavitation is based on combination of the Volume-of-Fluid technique (VOF) with an additional model for the growth and collapse of bubbles [4]. To model the turbulence effect, a k-ω model [5] is introuduced for the cavitationg flow. A finite-volume approach is used for the numerical discretization. Several complex cavitation phenomena are investigated using the present method.

THE MATHMATICAL MODEL AND NUMERICAL SOLUTION

Turbulence modeling

As pointed out in previous work [6], at present no appropriate turbulence models are available for addressing the dispersed two-phase dynamics involved in this flow field. As a preliminary investigation, we apply the Wilcox k-ω model [5] of sinle phase flows to the simulaton of cavitationg flows:
Dynamic eddy viscosity μ_t:

$$\mu_t = \frac{\rho k}{\omega} \tag{1}$$

Turbulence kinetic energy k:

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho \alpha x_j)}{\partial x_j}=\tau_{ij}\frac{\partial(\rho c_i)}{\partial x_j}-\partial\beta_k k\omega+\frac{\partial}{\partial x_j}\left[(\mu+\sigma_t\mu_t)\frac{\partial k}{\partial x_j}\right] \tag{2}$$

Specific dissipation rate ω:

$$\frac{\partial(\rho\omega)}{\partial t}+\frac{\partial(\rho\alpha x_j)}{\partial x_j}=\gamma\frac{\omega}{k}\tau_{ij}\frac{\partial(\rho c_i)}{\partial x_j}-\partial\beta_\omega k\omega^2+\frac{\partial}{\partial x_j}\left[(\mu+\sigma_\omega\mu_t)\frac{\partial\omega}{\partial x_j}\right] \tag{3}$$

where the indices are i =1,2 and j =1,2 for 2-Dflow problems. The specific Reynolds stress are given by

$$\tau_{ij} = -\frac{2}{3}\delta_{ij}\rho k + \mu_t\left(\frac{\partial\bar{c}_i}{\partial x_j}+\frac{\partial\bar{c}_j}{\partial x_i}\right) \tag{4}$$

Where the Kronecker symbol $\delta_{ij}=1$ for i=j. The clouser cofficents used in this study are listed in Table 1.

Tab le 1 Clouser cofficients for turbulence modeling

γ	β_ω	β_k	σ_ω	σ_k
0.52	0.072	0.09	0.5	0.5

Bubble Two Phase (BTP) model

The BTP model was used to consider interactions between viscous effects including vortices and cavitation bubbles [7, 8]. Here, inner and outer portion of a cavity are assumed as a single continuum by regarding the cavity as a compressible viscous fluid. This model treats the flow field macroscopically as a compressible viscous fluid whose density varies greatly. Microscopically, it is treated as cavitation structure like bubble clusters.

The BTP model uses a finite difference method to solve a system of partial differential equations including the 3-D Navier-Stoke`s and the Rayleigh`s equation. Here, P_{crit} is determined by the classical bubble dynamics approach, with the objective of determining pressure and velocity fields in a two-phase flow. In addition, the motions of the bubble wall are under the influence of time dependent pressure. The problem is complicated due to the coupling between the two phases.
Base on Blake`s threshold hypothesis for bubble nuclei, bubble equilibrium is solved in the BTP model using,

$$P_{crit} = P_v - \frac{4\sigma}{3R} \tag{5}$$

In the BTP model, the liquid is treated as incompressible. The mixture`s compressibility is attributed completely to the change in void fraction. The model assumes that a fluid of variable density replaces the liquid-vapor mixture. Due to very small vapor phase, the model neglects mass and momentum of the vapor. Only density variations due to phase change are considered. The previously stated relation between density and VoF, for the Barotropic model is applicable to the BTP model as well. However, additional relations for the viscous effect and local volume fraction are defined. Similar to the mixture density, the mixture viscosity is given by,

$$u = u_l C + u_v(1-C) \tag{6}$$

The simplified Local Homogenous Model (LHM), based on Mean Field Approximation, is used to define the local volume fraction by,

$$f_{local} = n\frac{4}{3}\pi\sum R^3 \tag{7}$$

Hence, the local vapor fraction is given by

$$C^* = 1 - n\frac{4}{3}\pi \sum R^3 \quad (8)$$

Assumptions made in deriving the simplified relations are; bubbles are always considered to be spherical, bubbles remain detached, value of n is assumed to be constant and bubbles are convected at liquid velocity i.e., a no slip condition exists.
Transport equation for the volume fraction, is similar to the one used for the Barotropic model except that an additional term for local cavitation source appears. This is given by,

$$\frac{\partial C}{\partial t} + \vec{U}\nabla C = \frac{C^*}{\partial t} \quad (9)$$

Rayleigh equation of radial motion, equation 10, governs the growth and collapse of the spherical bubbles.

$$R\frac{d^2R}{dt^2} + \frac{3}{2}\left(\frac{dR}{dt}\right)^2 = \frac{P_v - P}{\rho_L} \quad (10)$$

P_v is considered as a constant term in the Rayleigh equation for this model. This consideration explains the model`s isothermal nature. The BTP model further takes into account effects of surrounding bubbles on the bubble dynamics problem. Further, the BTP model treats cavitation microscopically as bubble clusters. In order to reduce the effect of the computational grid, the model considers interaction between individual bubbles. Modeling of the interaction is based on a Sub-Grid-Scale model. After treating equation 10 for bubble interactions, the final form of the Rayleigh-Plesset equation is;

$$\left(1 + 2\pi\Delta r^2 nR\right)R\frac{D^2R}{Dt^2}\left(\frac{3}{2} + 4\pi\Delta r^2 nR\right)\left(\frac{DR}{Dt}\right)^2 + 2\pi\Delta r^2\frac{Dn}{Dt}R^2\frac{DR}{Dt} = \frac{P_v - P}{\rho_l} \quad (11)$$

Equation 11 is the generalized form of the Rayleigh-Plesset equation, where as in case of BTP model, n is a constant. Also, in equation 11, the time derivate terms are replaced by total derivative terms to model the no-slip conditions.

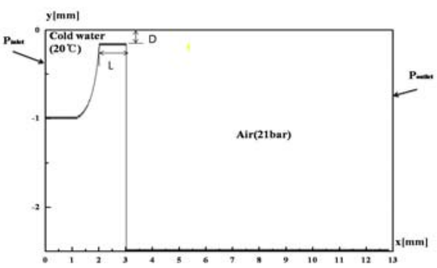

Fig. 3 The 2-D plane model for injection nozzle

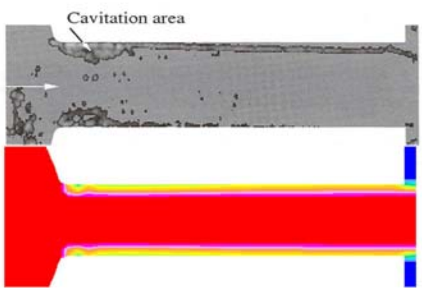

Fig. 4 The comparison of cavitating area between experiment and simulation

COMPUTATATINAL RESULTS & DISCUSSION

Input Conditions

As in previous work [9], two-dimensional plane experimental test results of Roosen et al. have been re-calculated to validate our numerical model and to study physical effects concerning the interaction between the cavitating flow and the external jet of the injection nozzle. The fluid used in the experimental was tap water. The nozzle hole is rectangular-shaped with dimension of 0.2 x 0.28 x 1 mm^3. In order to reduce the calculation time, a 2-D flows and symmetry with respect to the nozzle axis is assumed, see Fig.3. The computational mesh was generated about 180,000cells.
The computed results using the nuclei radius R_0=5μm and nuclei concentration n_0= 1 x 10^{14} 1/m^3water for the cases with $P_{injection}$=80bar, P_{exit}=21bar well agree with the experimental observations of Roosen et al, see Fig.4.
Further, spray characteristics and cavitating flow were investigated according to various aspect ratios of nozzle. These variables are listed in Table.2.

Table 2. Definition for various aspect ratio of nozzle

L/D	D(L=1)[mm]	L(D=0.28)[mm]
3	0.33	0.84
4	0.25	1.12
5	0.20	1.40
6	0.17	1.68
7	0.14	1.96

Effect of Aspect Ratios

According to the experiences of Chaves et al. [10], a cavitation was influenced by a nozzle diameter. The cavitation produces an increase in the spray angle, and this increase conforms to the most severe cavtitation.

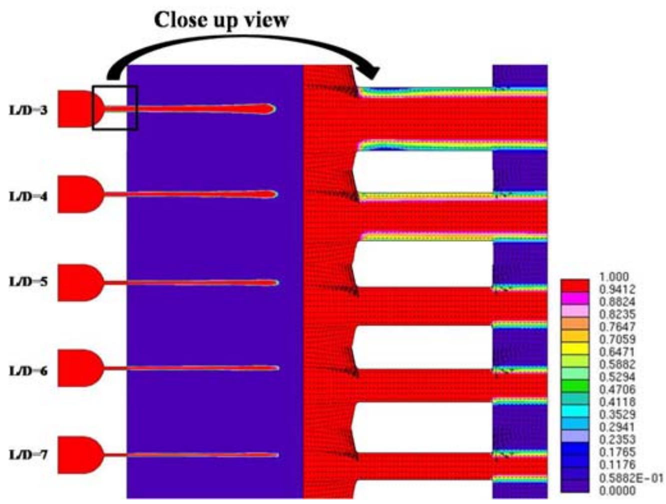

Fig. 5 Computed Volume-of-Fluid(VOF) and cavitation distribution as function of nozzle diameter at constant nozzle length(L=1mm)

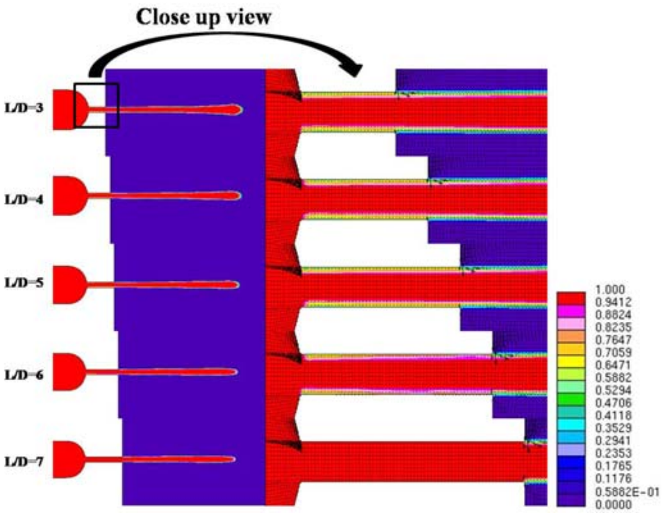

Fig. 6 Computed Volume-of-Fluid (VOF) and cavitation distribution as function of nozzle length at constant nozzle diameter (D=0.28mm)

Fig. 5 showed computed Volume-of-Fluid (VOF) and cavitation distributions as nozzle diameter changed. Here, spray penetration make no difference as aspect ratio increases. In all cases, length of the spray penetration was about 7mm when time is 0.001s. But, spray angle and cavitation size was larger than others, when aspect ratio was 3. When aspect ratio was 5, there isn`t cavitation flow where spray angle was reduced. Since it wasn`t influenced by cavitation flow, spray formation is shown as straight line type. Hence, increased nozzle diameter, produce cavitation, which could influence spray characteristics strongly.

Fig. 6 had shown computed Volume-of-Fluid (VOF) and cavitation distributions as nozzle length changed. Here, length of spray penetration and caviation size was reduced in respond to reduction of nozzle length. Cavitation disappeared when aspect ratio reached 7. Because it make disturbance in nozzle, non-cavitation occurs. As nozzle length varies, injection velocity was influenced by friction force between wall surface in the nozzle and fluid.

CONCLUSIONS

From the studies develop in this article we can draw the following conclusions;

(1) Cavitation affects spray formation including spray angle and length of spray penetration. Specially, aspect ratio of 3 for variation nozzle diameter showed large spray angle and cavitations. But, cavitations after point of critical aspect ratio disappeared after point of critical aspect ratio and spray formation was indicated straight line. Thus, cavitation was influenced strongly by nozzle diameter.

(2) As nozzle length increases, cavitation formation is reduced. Ultimately, aspect ratio of 7, cavitation disappeared and spray formation showed such as previous data. Moreover, according to increase aspect ratio in nozzle, length of penetration for spray showed least of others

(3) Cavitation and spray behaviour showed clearly, when aspect ratio is 3 and 4.

(4) Spray formation was strongly influenced by cavitation.

ACKNOWLEDGMENT

This work was supported by the second phase of the Brain Korea 21 program in 2008 and the Clean Coal Center, Pusan National University, under the program of University Electric Power Research Center, Ministry of Knowledge and Economy, Korea. I would like to thankful for them.

NOMENCLATURE

Symbol	Quantity	SI Unit
μ_t	Dynamic eddy viscosity	m/s
k	Turbulence kinetic energy	
ω	Specific dissipation rate	
τ	Specific Reynolds stress	
P_{crit}	Pressure at which cavitation takes place	Pa
P_v	Vapor Pressure of fuel	Pa
ρ_v	Vapor density	Kg/m^3
C	Volume ratio of liquid	
(1-C)	Volume ratio of vapor	
$\vec{U}$	Velocity vector	m/s
μ_l	Fuel viscosity	Kg/m·s
μ_v	Vapor viscosity	Kg/m·s
f_{local}	Local void fraction	
n	Bubble number density	
C^*	Local void fraction	
C_k	Model constant	

REFERENCES

[1] Arcoumanis C., Plensdorf W., Lavoie G,, Curtis E., The Occurene of Flash Boiling in a Port Injected Gasoline Engine,SAE Paper, 970799, 1998.

[2] Baumgarten C., Stegemann J., Merker GP., A New Model for Cavitation Induced Primary Breakup of Diesel Sprays, Proc 18th ILASS Europe conf, pp15~20, 2002.

[3] Bergwerk W., Flow pattern in diesel nozzle spray holes, Proceedings of the Institute on Mechanical Engineers 173, pp655~660, 1959.

[4] Rayleigh L., On the Pressure Developed during the Collapse of a Spherical Cavity, Phil. Mag. S. 6 34(200), pp94~98.1917.

[5] Wilcox D.C., Turbulence Modeling for CFD, CDW Industries, Inc., La Canada , CA, 1998

[6] Yuan W., Sauter J., Schnerr G.H., Modeling of Unsteady Cavitating Flows in Fuel Injection Nozzles, ILASS-Europe 2000, pp11~13, 2000

[7] STAR-CD, Methodology Volume, 1999.

[8] Kubota A., Kato H. Yamaguchi H., A Numercal Study of Unsteady Cavitation on a Hydrofoil Section, Jornal of Fluid Mechanics 240,pp59~96, 1991.

[9] Yuan W., Sauter J., Schnerr G.H., Modeling and Computation of Unsteady Cavitation Flows in Injection Nozzle, Mecanique & Industries 2, pp383~394 , 2001.

[10] Chaves H., Knapp M., Kubitzek A, Experimental study of cavitation in the nozzle hole of diesel injectors using transparent nozzles,SAE paper950290,1995.