International Communications in Heat and Mass Transfer 59 (2014) 1–10

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

journal homepage: www.elsevier.com/locate/ichmt

Experimental study of flow regime characteristics in diesel multi-hole nozzles with different structures and enlarged scales ☆

Wenjun Zhong, Zhixia He *, Qian Wang, Zhuang Shao, Xicheng Tao

School of Energy and Power Engineering, Jiangsu University, Zhenjiang 212013, China

ARTICLE INFO

Available online 12 October 2014

Keywords:
Diesel
Nozzle structure
Flow regime
Cavitation
Visualization experiment
Spray

ABSTRACT

The internal flow of diesel nozzles has a strong influence on the subsequent spray and atomization characteristics. A flow visualization experiment system equipped with enlarged transparent nozzles was set up to investigate the effect of different nozzle structures and different nozzle enlarged scales on cavitating flow and subsequent spray characteristics. Significant parameters of internal flow including critical pressure of cavitation inception, critical pressure of hydraulic flip, discharge coefficient, Reynolds number, flow rate and cavitation distribution were fully investigated during the experiments. Experimental results show that the nozzles with small length/diameter ratios at the same orifice diameter were more inclined to cavitate and had higher discharge coefficient than that of nozzles with large length/diameter ratios. Cavitating flow in nozzles with different sac volume structures may incur different spray characteristics in the near-nozzle field and it is the internal cavitating flow that induces the asymmetry of subsequent spray. The investigations indicate that the critical pressure of hydraulic flip and discharge coefficient of nozzles under different needle lifts were quite different even under the same boundary conditions and the nozzle flow characteristics have been greatly influenced by the hydraulic flip phenomenon. Two types of cavitation were observed in nozzles with different scaled-up times and the string cavitation was found in three times scaled-up nozzle, while the cloud cavitation bubbles were appeared in five and eight times scaled-up nozzles. A kind of hysteretic cavitation phenomenon was also observed in the experiment and then was analyzed in this paper.

1. Introduction

The injector nozzle is one of the most important parts of a Diesel engine and uses high injection pressures to form a fuel spray of small droplets at very high speeds. The nozzle has then much to do with the atomization process and spray behavior and can be decisive for engine performance and pollutant formation [1,2]. In order to improve the quality of diesel spray, the electronically controlled high-pressure common-rail fuel injection system [3,4] has been introduced into the market for controlling injection quantity and injection time more accurately. In this system, the fuel injection pressure can be over 200 MPa and the cavitating flow inside the nozzle hole is unavoidable. It is this two-phase cavitating flow extending to the exit of the nozzle hole that induces the flow disturbance and liquid velocity turbulence on the surface of the liquid jet and then has been identified to be one of the most important factors influencing the fuel atomization [5–9]. So during the past years, many researchers have put focus on the nozzle internal cavitating flow regime characteristics and its effect on the subsequent spray behavior.

Cavitation inception and its effect on the spray can be attributed to the two kinds of factors, "geometrical" and "dynamic" factors [10]. Geometrical parameters include nozzle orifice length/diameter ratios, the type of nozzle sac volume (valve covered orifice (VCO) or mini-sac), orifice inlet curvature, and orifice inclination angle. Dynamic parameters include the injection pressure, injector needle lift and needle eccentricity. In terms of geometrical factors, the effects of nozzle structures on cavitating flow have been widely carried out through CFD method by Payri et al. [11], Sou et al. [12], Reitz et al. [13], Som et al. [14], reported the effects of nozzle orifice length, nozzle inlet radius and nozzle passage convergence on discharge coefficients. It was found that the rounded-inlet nozzles tend to reduce cavitation and have larger discharge coefficients compared to sharp-inlet nozzles. Som et al. [14] compared the effects of conical and cylindrical orifices of nozzles on internal flow, spray processes, combustion and emission characteristics by coupling the injector flow and spray in CFD software CONVERGE. While experimental studies will still be more important for revealing the cavitating flow mechanisms and flow regime characteristics in nozzles with different orifice geometries. Soteriou et al. [9] studied the internal flow structures in a scaled-up plain orifice nozzle and observed cavitation inception at three distinct locations, namely a separated boundary layer inner region, a main stream flow region and an attached boundary layer inner region. Payri et al. [15,16] reported the effect of

☆ Communicated by W.J. Minkowycz.
* Corresponding author.
E-mail address: zxhe@ujs.edu.cn (Z. He).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2014.10.001

Nomenclature

A	cross-sectional area of a nozzle hole, m^2
C_d	discharge coefficient
D	nozzle hole diameter, mm
H	needle lift, mm
L	nozzle hole length, mm
K	cavitation number
Q_v	volume flow rate, m^3/h
$\dot{m}$	actual mass flow rate of a nozzle, kg/s
Re	Reynolds number
P	fluid pressure, Pa
P_v	vapor pressure, Pa

Greek letters

ρ	mixture density, kg/m^3

subscript

crit	critical parameter
l	liquid phase
in	upstream location of a nozzle
back	nozzle hole outlet

cylindrical nozzle and convergent nozzle on cavitating flow, and measured the spray momentum flux and mass flux in order to explain the effects of cavitation on diesel spray behavior. Lee et al. [17] investigated the cavitating flow in a tapered nozzle and found that cavitation could not be formed in a tapered nozzle even with a much higher injection pressure. Suh et al. [18] considered the effects of orifice different length/diameter ratios on cavitating flow and fuel atomization characteristics using a simplified transparent planar nozzle with one hole. From Suh's experimental results, the cavitation generated in the nozzle enhances the fuel atomization performance and the longer nozzle orifice length induces better fuel atomization. The nozzle orifice length/diameter ratios, as one of the most important nozzle geometrical parameters, need to be further investigated in the experiment with the multi-hole nozzle but not just with the planar or single-hole nozzle.

Additionally, nozzle sac volume structure has been proved to be another important nozzle geometrical factor to greatly affect the spray behavior [19], while its effect on nozzle internal flow is not clear enough as yet. He et al. [20] studied the effects of the nozzle sac volume, orifice inlet curvature, orifice inclination angle, injector needle lift and needle eccentricity on the internal cavitating flow of nozzles by CFD method. The results concluded that the flow characteristics of the improved nozzle can be better than that of the VCO nozzle and STD nozzle. Yoshinaka et al. [21] investigated the effects of straight step hole (VCO-S) and tapered step hole (VCO-T) nozzles on the NOx emissions and spray penetration. It concluded that the VCO-S nozzle could reduce NOx emissions more than the VCO-T nozzle. And its spray penetration was shorter than that of the VCO-T nozzle. Some researchers studied the effects of different sac volume structures of nozzles on emissions and spray characteristics. However, the influences of sac volume structures on nozzle cavitating flow are rarely studied through experiment method, which also need to be further performed.

For the dynamic factors, the injector needle lift is one of the important parameters to affect the cavitation inception, cavitation development and subsequent spray, but the investigations on the nozzle cavitating flow considering of the injector needle movement are very limited, especially by experimental method. Payri et al. [22] numerically investigated the effect of needle movement on the cavitation pattern of diesel injectors. Results show that the cavitation pattern varies larger throughout the nozzle with movement of the needle and then induces quite different flow characteristics at the nozzle exit. Salvador et al. [23] and He et al. [24] studied the influence of needle lift on the internal flow also by CFD method with RANS model. Salvador concluded that the turbulence development depends on the needle lift and is strongly related to pressure evolution in the nozzle. He's results show that the cavitation distribution is quite different during the period of needle opening and closing. Although numerical simulation studies have revealed that the cavitation patterns are quite different under different needle lifts, while experimental investigation concerning about the flow regime, critical pressure of cavitation inception and critical pressure of hydraulic flip are quiet insufficient, which need to be further studied.

Experimental study of nozzle cavitating flow has long been paid more attention. Due to the extremely small geometry of the nozzle, whose diameter ranges from about 0.1 mm to 0.3 mm and length is about 1 mm for most automotive diesel engine. Hence, most of researches were performed with large-scale transparent nozzles to obtain useful information. Arcoumanis et al. [25] observed the cavitating flow in an enlarged transparent diesel injector with a vertical multi-hole model. The experimental results revealed that the flow development in multi-hole vertical nozzles, despite the axisymmetric geometry, may lead to hole-to-hole flow variations as a result of the transient nature of the cavitation structures formed inside the sac volume and holes. Gavaises et al. [26] concluded that the cavitation strings are formed in the areas where large vortical structures are present. Sou et al. [27] designed a simple scaled up nozzle to understand how it forms and affects the atomization of the injected liquid jet. Due to the cavitation structures are very different in nozzles with different enlarged scales [28,29]. Hence, it is very important to analyze the effect of different scales of nozzles on internal cavitating flow. While the literature and the studies on this subject are rather poor.

In this paper, a high speed imaging visualization system equipped with enlarged transparent nozzles was established. The effect of different orifice length/diameter ratios (L/D), sac volume structures, needle lifts and enlargement factors on the internal cavitating flow and its subsequent spray characteristics were fully investigated through experiments. A large number of images, experimental data and important parameters including critical pressure of cavitation inception, critical pressure of hydraulic flip, cavitation structures, cavitation number (K), discharge coefficient (C_d), flow rate (Q_v) and Reynolds number (R_e) were all presented and analyzed in this paper.

2. Parameter descriptions

Fundamentally, the transition of liquid to vapor can be achieved by heating the fluid at a constant pressure, known as boiling, or by decreasing the pressure at a constant temperature, which is known as cavitation. For most applications, cavitation is hypothesized to occur as soon as the local pressure of working fluid drops below the vapor pressure of the fluid at a specified temperature. In diesel injector nozzles, due to the abrupt change of flow direction at the nozzle orifice inlet, the boundary layer tends to separate from the wall at the inlet section. As a consequence, a recirculation flow accompanied with a pressure drop due to the accelerations motion of working fluid appears in this zone.

For investigating the nozzle cavitation characteristics, two non-dimensional parameters are worth to be mentioned, one is discharge coefficient, and the other is cavitation number.

The discharge coefficient of a nozzle can be obtained by combining the Bernoulli equation and the mass conservation equation:

$$C_d = \frac{\dot{m}}{A\sqrt{2\rho_l(p_{in} - p_{back})}} \tag{1}$$

where $\dot{m}$ is the actual mass flow rate, A is the cross-sectional area of a nozzle hole, ρ_l is the liquid density, P_{in} is the upstream pressure (injection pressure) and P_{back} is the orifice outlet pressure.