International Journal of Heat and Mass Transfer 104 (2017) 18–27

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

Numerical analyses of transient flow characteristics within each nozzle hole of an asymmetric diesel injector

Fuying Xue [a], Fuqiang Luo [a,*], Huifeng Cui [a], Adams Moro [a,b], Liying Zhou [a,c]

[a] School of Automotive and Traffic Engineering, Jiangsu University, Zhenjiang 212013, China
[b] Mechanical Engineering Department, Accra Polytechnic, Accra, Ghana
[c] School of Mechanical Engineering, Guiyang University, Guiyang 550005, China

ARTICLE INFO

Article history:
Received 23 March 2016
Received in revised form 27 July 2016
Accepted 10 August 2016

Keywords:
Off-road diesel engine
Asymmetric multi-hole injector
Each nozzle hole
Flow characteristics
Numerical analyses

ABSTRACT

In order to simultaneously study the differences in transient flow characteristics within each nozzle hole of an asymmetric multi-hole diesel injector used in off-road machinery, a three-dimensional gas–liquid two-phase flow model of cavitation was developed, which takes the influence of injection conditions and bubble number density into consideration. The computational model was validated by comparing it with the experiment conducted by Winklhofer. The result shows high level of consistency and sensitivity. Following the successful verification of the model, the differences in transient flow characteristics within each nozzle hole based on a P-type asymmetric multi-hole nozzle with sac were analyzed using the model. Results obtained from simulation shows that the cavitation phenomenon, velocity profile and mass flow rate in each nozzle hole of an asymmetric injector differ greatly. The hole with the higher nozzle hole angle (wider angle between nozzle hole axis and needle axis) was more inclined to cavitate, therefore the cavitation effect intensity is directly related to the size of the nozzle hole angle. During the lifting stages of the needle valve, the cavitation effect for each nozzle hole and the fuel flow velocity are gradually enhanced while on the contrary, the fuel flow velocity decreases and cavitation effect slightly increases at the closing stages of the needle valve lift. Also, the cavitation morphology at the same needle valve lift in any nozzle hole differs during opening and closing stages of fuel injection. In addition, the fuel flow velocity profile from the needle valve opening at the sac and into the hole of each nozzle shows that, in the order of increasing nozzle hole angle β_i, the velocity effect gradually reduces as a result of dramatic change in flow direction.

1. Introduction

Off-road vehicular machinery equipped with diesel engines have attracted global attention with substandard emission quality which is widely known to be caused by inefficient combustion processes. Combustion processes are directly affected by the quality of air–fuel mixtures obtained from spray processes of fuel injection system [1,2]. Reviews of previous research works point to the fact that the fluid flow regime, turbulence and cavitation, present in the injection nozzle holes plays an important role in the fuel atomization and significantly affects spray characteristics, such as spray penetration, Sauter Mean Diameter (SMD) and fuel distribution [3–8]. Soteriou et al. [6] and Hiroyasu et al. [7] discovered that the cavitation within injection nozzle holes have significant effect on the primary characteristics of the atomization of jet flow. Specifically, cavitation effect enlarges the spray cone angle and shortens the break-up length of jet flow. The atomization process and quality of air–fuel mixture are influenced by the jet rub against surrounding air and fluid flow characteristics inside the nozzle hole. The existence of cavitation and the turbulence flow through the nozzle holes are vital factors necessary for jet atomization.

Investigation to better understand the transient flow characteristics within nozzle holes is therefore essential given the effect it has on spray process in determining various engine performances. Comprehensive experimental and numerical simulation studies have been carried out by scholars both at home and abroad [9–14]. Some researchers study the effect of geometrical factors (orifice length, orifice diameter, the ratio of orifice length to diameter, the inlet curves' radius, cylindrical nozzle, conical nozzle, elliptical nozzle, the nozzle sac volume, orifice inclination angle) on the internal flow within nozzle hole [9,13,15–17]. Molina et al. [17] studied the influence of employing elliptical orifices on the nozzle inner flow and cavitation development, and pointed

* Corresponding author.
E-mail address: luofq@ujs.edu.cn (F. Luo).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2016.08.027
0017-9310/

Nomenclature

A_i'''	interfacial area density (1/m)
A_{geo}	geometrical outlet section (m^2)
C_{TD}	turbulent dispersion coefficient (dimensionless)
C_d	discharge coefficient (dimensionless)
C_D	drag coefficient (dimensionless)
D_b	bubble diameter (m)
F^D	drag force (N)
g	acceleration of gravity (m/s^2)
k	turbulence kinetic energy (m^2/s^2)
M	interfacial momentum transfer (kg/s^2m^2)
$\dot{m}$	mass flow rate (kg/s)
N'''	bubble number density (#/m^3)
P_i	injection pressure (Pa)
P_o	outlet pressure (Pa)
P_v	vapor pressure (Pa)
R	mean bubble radius (m)
T^t	Reynolds stress (kg/s^2m)
t	Time (s)
$\boldsymbol{v}$	velocity vector (m/s)
v	kinematic viscosity (m^2/s)
V	flow velocity (m/s)

Greek symbols

∇	laplace operator
ΔP	effective differential pressure (Pa)
α	volume fraction (dimensionless)
ε	turbulence dissipation rate (m^2/s^3)
β	angle between nozzle hole axis and needle axis (°)
τ	shear stress (kg/s^2m)
ρ	liquid density (kg/m^3)
Γ	mass exchange term (kg/m^3s)

Subscripts

i	nozzle hole serial number
k, l	phase index

out that elliptical geometries with vertically oriented major axis are less prone to cavitate and have a lower discharge coefficient, whereas elliptical geometries with horizontally oriented major axis possess the opposite effect.

Other researchers focus on the effect of dynamic factors (injection pressure, the ratio of injection pressure to the ambient pressure, injection pressure fluctuations, needle movement, and needle eccentricity) on the internal flow within nozzle holes [18–20]. Salvador et al. [19] made an in-depth study on nozzle holes inner flow at different needle valve lifts, and discovered that at various needle valve lifts, the influence of cavitation effect on the main flow features differs. Wang et al. [20] studied the relationship between injection pressure fluctuations and cavitation processes and they established that the evolution of cavitation bubbles in recirculation zone and its wake flow responds differently to the time derivatives of the upstream pressure.

In addition, some researchers have also explored the internal flow characteristics within nozzle hole from the perspective of fuel properties (liquid viscosity, liquid temperature, different fuel type: diesel and biodiesel) [21–24]. Giorgi et al. [22] established that flow temperature influences both cavitation intensity and cavitation number at which different two-phase flow regime transitions occurs, and this increases with increasing flow temperature. Som et al. [24] pointed out that due to the lower vapor pressure of biodiesel, the cavitation and turbulence levels of biodiesel are significantly lower compared with the conventional diesel.

Although most studies focus on cavitation effect (in single or symmetric nozzle) with regards to geometric and flow parameter, the cavitation morphological effect (for asymmetric nozzle) at the same needle valve position during both opening and closing stages of the fuel injection system has not been thoroughly investigated. Errors in manufacturing process during the production of injectors have been associated to some degree as being the cause of injection rate differences within symmetric multi-hole nozzles and therefore directly affect the hydraulic characteristics of fluid flowing through the nozzles [25]. Hence contributing to the non-homogeneous distribution of fuel in space and time inside the combustion chamber, which leads to imbalance of thermal load, deterioration of combustion and emission quality [25–28].

Owing to the challenging task in studying transient flow characteristics of multi-hole nozzles experimentally (especially for asymmetric multi-hole nozzle), numerical computation methods are used significantly nowadays in analyzing the flow regimes. In order to explore the characteristics of transient flow parameters within each nozzle hole of an asymmetric multi-hole diesel injector used in off-road machinery, a three-dimensional gas–liquid two-phase flow cavitation model was developed, which takes the influence of injection conditions on bubble number density into account. Most Internal combustion engine flows are turbulent, especially in-nozzle flow of diesel injectors and therefore to ensure the adequate determination of the physical processes, turbulent phenomenon was accurately replicated in the geometric model.

In Section 2 of this paper, a brief introduction of the mathematical theory associated with the cavitation and turbulence model is presented. The description of the computational model and the validation process are in Section 3 while in Section 4, computational study results on transient flow characteristics differences among nozzle holes of an asymmetric multi-hole diesel injector are offered. Finally, the salient conclusions are presented in Section 5.

2. Mathematical models

Cavitation flow within nozzle holes are modeled using homogeneous flow approach [29,30], volume-of-fluid (VOF) method [31] and two-fluid model approach [32–35]. Comparatively, the two-fluid model approach gives detail description of flow with less amount of computational time [33]. As a result, the two-fluid model approach implemented in the AVL FIRE commercial code was used for the simulation to replicate turbulence flow characteristics within the geometric model of the diesel injection system.

For incompressible flow, the divergence-free velocity is:

$$\nabla \cdot \boldsymbol{v} = 0 \tag{1}$$

where ∇ is the Laplace operator, the $\boldsymbol{v}$ is the velocity vector of the flow in space.

The mass and momentum conservation equations of the two-fluid method approach are:

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k = \sum_{l=1, l\neq k}^{2} \Gamma_{kl} \tag{2}$$

$$\frac{\partial \alpha_k \rho_k v_k}{\partial t} + \nabla \cdot \alpha_k \rho_k v_k v_k = -\alpha_k \nabla p + \nabla \cdot \alpha_k (\tau_k + T_k^t) + \alpha_k \rho_k g + \sum_{l=1, l\neq k}^{2} M_{kl} + v_k \sum_{l=1, l\neq k}^{2} \Gamma_{kl} \tag{3}$$