International Journal of Heat and Fluid Flow 59 (2016) 1–9

Contents lists available at ScienceDirect

International Journal of Heat and Fluid Flow

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

Numerical research of diesel spray and atomization coupled cavitation by Large Eddy Simulation (LES) under high injection pressure

B. Yin[a,*], S. Yu[a], H. Jia[a], J. Yu[b]

[a] School of Automobile and Traffic Engineering, Jiangsu University, Zhenjiang 212013, China
[b] Wuxi Fuel Injection Equipment Research Institute, Wuxi 214063, China

ARTICLE INFO

Article history:
Received 11 November 2015
Revised 14 January 2016
Accepted 26 January 2016
Available online 17 February 2016

Keywords:
Diesel nozzle
Cavitation
Spray and atomization
Large eddy simulation

ABSTRACT

A special spray model is applied to study the spray behavior with high injection pressure and micro-hole nozzle. To reveal the cavitation in diesel nozzle and its influence on spray and atomization, the Large Eddy Simulation (LES) turbulence model is adopted to detect the cavitation, and then the special spray model coupling the cavitation is build. From research results, three important conclusions can be drawn. Firstly, the cavitation flow can raise the effective velocity at the nozzle exit and such effect become even more obvious with higher injection pressure, e.g.180 MPa. Secondly, the applied spray model is in good agreement with the spray characteristics and images obtained from the EFS8400 spray test platform. Thirdly, the cavitation with high injection pressure and micro-hole nozzle can increase the spray cone angle and reduce the spray penetration; the cavitation intensity has a great impact on the spray velocity field and vorticity intensity, especially at the initial spray field under the condition of high injection pressure.

1. Introduction

In direct injection diesel engines, the quality of atomization greatly affects the fuel consumption and emissions of engine. The increasingly stringent emission regulations e.g. Europe V and focus on environmental pollutions have put more requirements on the fuel injection system. Injection pressure and nozzle diameter, referred to as the main two factors in a fuel injection system (Suh et al., 2007), presents a trend of high injection pressure ($>$180 Mpa) and micro-hole diameter ($<$120 μm). In that case, the mechanisms of cavitation and the characteristics of spray and atomization show great complexities in comparison to traditional injection pressure and diameter.

Tamaki et al. and Hiroyasu (Hiroyasu, 2000; Tamaki et al., 2001; Tamaki et al., 1998) found that the disturbance phenomenon inside the nozzle was caused mainly by cavitation. The cavitation bubbles broken at the nozzle exit could enhance the formation of micro jets and fuel atomization. It was also found that although the fluid pressure dropped substantially across micro nozzle, the atomization quality was still poor if the cavitation phenomenon did not appear in the nozzle. Nurick (1976) observed that cavitation inside the nozzle could reduce the quality of mixture formation. Ruiz (He and Ruiz, 1995) conducted research on the influence of cavitation in which the nozzle velocity fields were measured under conditions of both cavitation and non-cavitation. It was presented that the turbulence intensity in cavitation flow decayed was faster than that in non- cavitation conditions. Soteriou et al. (2001) emphasized the importance of cavitation for primary spray. He found that cavitation greatly increased the spray cone angle and shortened the jet breaking length, improving the quality of atomization. Wu et al. (1983) stated that the spray cone angle under the cavitation flow nozzle was bigger than that under the non-cavitation flow conditions through experimental data. Chaves et al. (1995) found that cavitation could increase the spray cone angle. This was mainly due to the fact that broken cavitation bubbles reinforced the turbulence at the nozzle exit. Ganippa et al. (2004) presented that the growth of cavitation region inside the nozzle could raise the nozzle exit effective velocity. This was merely because the cavitation region attached to the nozzle wall could reduce the friction between the wall and fluid, thus accelerating the mass flow.

Some studies concluded that the appearance of cavitation inside the nozzle had a great impact on the spray and atomization. Owing to huge difficulties in the visualization of cavitation and spray characteristics under the high injection pressure and micro-hole nozzle (Giannadakis et al., 2004; Payri et al., 2005; Soteriou et al., 1999), CFD tool has become an effective method when studying the mechanisms of the cavitation together with spray and atomization.

---

[*] Corresponding author. Tel.: +8613852910199.
E-mail address: ybf@ujs.edu.cn (B. Yin).

http://dx.doi.org/10.1016/j.ijheatfluidflow.2016.01.005
S0142-727X(16)30002-9/

Nomenclature

B_0	empirical constant, =0.61
B_2	empirical constant, =1
D	nozzle diameter, mm
D_{eff}	effective nozzle diameter, mm
K	von Kármán constant
L_s	mixing length for sub grid scales
m	mass flow rate, g/s
P_B	bubble surface pressure, Pa
P	local far-field pressure, MPa
P_{inj}	injection pressure, MPa
P_v	liquid vapor pressure, Pa
R_e	vapor generation rate
R_c	vapor collapse rate
$\Re_B$	bubble radius, mm
U	nozzle exit velocity, m/s
V_v	vapor phase velocity, m/s
ρ_v	vapor density, kg/m^3
ρ_l	liquid density, kg/m^3
μ_t	subgrid-scale turbulent viscosity, N.s/m^3
Ω	the biggest growth disturbulance
Λ	wave length, mm

Subscripts

B	bubble
c	collapse
e	generation
eff	effective
inj	injection
l	liquid phase
s	sub grid scales
t	turbulent viscosity
v	vapor phase

Sou et al. (2014) proposed a new combination of Large Eddy Simulation (LES), Eulerian–Lagrangian Bubble Tracking Method (BTM), and the Rayleigh–Plesset (RP) equation to examine the incipient and developing cavitation flows in fuel injector. Egerer et al. (2014) presented Large-eddy simulations (LES) of cavitating flow of a Diesel-fuel-like fluid in a generic throttle geometry, and the LES with the employed cavitation modeling predicts relevant flow and cavitation features accurately within the uncertainty range of the experiment. Salvador et al. (2013) studied the turbulence effects on cavitation by Large Eddy Simulation (LES), and indicate that the turbulence developed in the discharge orifices can enhanced the cavitation intensive.Reynolds-averaged Navier–Stokes method and homogeneous equilibrium models are adopted to investigate the cavitation flow in different elliptical and circular orifices by Molina et al. (2014). Som et al. (2010) reported a comprehensive investigation of internal nozzle flow characteristics and cavitation phenomenon inside a single orifice of diesel injector with a mixture model. Shervani-Tabar et al. (2013) studied the influence of injection pressure on the spray penetration, sauter mean diameter and the evaporation of the fuel characteristics by numerical simulation. The results showed that with the increase of injection pressure the spray penetration increased, SMD decreased, and the spray atomization quality improved. Payri et al. (2004) and Lee et al. (2002) studied the diesel spray characteristics considering cavitation through simulation. It was found that cavitation could decrease the spray penetration and the SMD. Also, the spray distribution was more uniform. Suh and Lee (2008) studied the influence of nozzle structure on cavitation and spray atomization by simulation method and found that the cavitation inside the nozzle became more fierce when the hole length to diameter was smaller, which is very helpful to improve the quality of primary spray atomization.

A good number of experts have made significant achievements in diesel spray behaviors through experimental and numerical methods and stated that the cavitation flow in diesel nozzle has a great impact on the spray and atomization (Kong et al., 2007; Mohan et al., 2014). However, the existing studies of spray and atomization do not fully consider the cavitation. The cavitation inside the nozzle becomes more intense and complicated when the injection pressure increases and the nozzle hole diameter diminishes. Furthermore the mechanisms of fuel spray and atomization show great complexities. Therefore, the traditional turbulence model and numerical simulation methods cannot adapt to the current situation any longer. There is necessity to deeply investigate the turbulence model and the mechanisms of spray and atomization when fully considering the cavitation.

In this work, to take cavitation into consideration, a new spray model is developed by inducing cavitation model into spray sub model WAVE. The newly-developed spray model is extensively validated against the experimental data (e.g. spray penetration, spray cone angle and spray images) obtained from EFS8400 spray test platform. Firstly, considering that the cavitation becomes more intense in micro-hole with high injection pressure, the LES turbulence model is proposed to comparatively study the development of cavitation with different injection pressures and the influence of cavitation on the nozzle discharge characteristics. Secondly, the Enhanced BLOB theory (von Kuensberg Sarre et al., 1999) is used for coupling the cavitation in spray model. Thirdly, the macro characteristics of spray and the velocity field and the vortex field of spray coupling cavitation and non-cavitation are deeply investigated under the condition of different injection pressures and nozzle hole diameters.

2. Numerical modeling

A commercial CFD code Fluent is used to perform the calculations in this study. The mixture model, which assumed that liquid and vapor phases were always mixed uniform, was selected to describe the multiphase flow. In order to save the computational cost and acquire more information under violent turbulence conditions, the LES turbulent model was applied in the cavitation simulations. PISO algorithm, highly recommended for transient calculations, was used to solve pressure equation and correct the velocity. In doing so, continuity can be achieved more quickly.

2.1. LES governing equation

The governing equations for flow field calculation are the continuity and momentum equations. The filter equations are Eqs. (1) and (2).

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho \overline{u_i}) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\rho \overline{u_i}) + \frac{\partial}{\partial x_j}\left(\rho \overline{u_i u_j}\right) = -\frac{\partial \overline{p}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u_i}}{\partial x_j}\right) - \frac{\partial \tau_{ij}}{\partial x_j} \tag{2}$$

The Smagorinsky–Lilly model (Payri et al., 2010, 2011; Piomelli, 1999) was used in this paper, where the SGS stress in Eq. (2) is defined as:

$$\tau_{ij} - \frac{1}{3}\tau_{kk}\delta_{ij} = -2\mu_t \overline{S_{ij}} \tag{3}$$

$$\overline{S_{ij}} = \frac{1}{2}\left(\frac{\partial \overline{u_i}}{\partial x_j} + \frac{\partial \overline{u_j}}{\partial x_i}\right) \tag{4}$$

One of the most appropriate methods to model the internal flow, Smagorinsky model, was used. Where μ_t is the sub grid-scale