International Journal of Heat and Mass Transfer 112 (2017) 18–27

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

LES investigations on effects of the residual bubble on the single hole diesel injector jet

Genmiao Guo [a], Zhixia He [b,*], Yuhang Chen [a], Qian Wang [a], Xianyin Leng [b], Shenxin Sun [a]

[a] School of Energy and Power Engineering, Jiangsu University, Zhenjiang, China
[b] Institute for Energy Research, Jiangsu University, Zhenjiang, China

ARTICLE INFO

Article history:
Received 23 August 2016
Received in revised form 10 March 2017
Accepted 18 April 2017

Keywords:
Diesel nozzle
Large eddy simulation
Residual bubble
Atomization
Cavitation

ABSTRACT

Bubbles surviving from the last injection process play an important role in the early stage of the next injection process, and ultimately affect the quality of diesel spray characteristics. In this paper, a combination of the LES (Large Eddy Simulation) method and VOF (Volume of Fluid) model was applied to study the unsteady liquid jet flow at the early stage of injection coupling the cavitating flow inside orifices, especially to analyze the effects of the residual bubble on the diesel jet. It was found that the instability of the jet surface was intensified by the cavitating flow inside the nozzle. The development of jet instability waves and the cavitation collapse inside the jet aggravated the interactions between air and diesel jet, which caused the liquid core to separate into some large liquid bulks a bit earlier. Then, the ligaments formed from the bulks broke up into droplets. Furthermore, the double mushroom-like shape occurred with the residual bubble, while just a single mushroom head occurred without the residual bubble. The length of liquid core decreased with the increasing of injection pressure, and the phenomenon that the air inside the spray chamber was partly sucked into orifice was observed.

1. Introduction

Requirements for the diesel injection system have become higher to satisfy the strict emission standards. The quality of spray during the injection period is mainly influenced by the atomization of the liquid jet, which plays an important role in the subsequent combustion process and the emissions [1]. The diesel jet spray process is a complicated multi-phase flow phenomenon involving heat transfer and phase change. Especially, the unsteady liquid jet at the early stage of the injection and its breakup in diesel injectors have recently become one of the most important research focuses [2–5].

The purpose of this study, on one hand, is understanding the cavitation phenomenon inside the nozzle orifice and its influence on jet breakup processes and, on the other hand, is analyzing the characteristics of the primary breakup by LES (Large Eddy Simulation) method, especially the effects of the residual bubble on the jet. According to the generation mechanisms, the cavitation in diesel fuel injection system can be clarified into dynamics-induced and geometry-induced cavitation [6,7]. Primary breakup occurring in the vicinity of the nozzle outlet is the initial process of atomization. Instabilities generated by primary breakup provide the initial conditions for secondary breakup in the disperse flow region and have a significant effect on the whole development of the spray process [5,8].

Despite the fact that the characteristics of cavitation inside the diesel nozzle orifice are widely discussed in many investigations [6,9–11], the influences of cavitation on the jet breakup process and characteristics of the subsequent spray are still not well predicted due to high injection pressure, small length and time scales of the fuel injection process. So far, many theories have been proposed to describe the primary atomization mechanism including cavitation-induced disintegration [12,13], aerodynamic shear forces [13], turbulence-induced disintegration [14,15], relaxation of the velocity profile [16], liquid bulk oscillation [13,17] and so on, but a consistent view has not been achieved. Many interdependent phenomena, such as turbulence in nozzles [14,15] which is strengthened by the collapse of cavitating bubbles [12,18], fluctuations of the injection velocity [16] and cloud cavitation shedding [19,20], can provoke severe velocity fluctuations at the exit of nozzle hole leading to a nonlinear instability of the primary breakup. Furthermore, bubbles surviving during the period between two injections play an important role in the next injection process. This is a challenging subject to study, both experimentally and numerically. Badock et al. [18] found that during the needle closing period, the dissolved diesel vapor and air occur in the sac and orifice in the form of large bubbles, then at the early stage of the next injec-

* Corresponding author.
E-mail address: zxhe@ujs.edu.cn (Z. He).

http://dx.doi.org/10.1016/j.ijheatmasstransfer.2017.04.080
0017-9310/

tion process the bubbles in the orifice were partly pushed out the orifice first. Jiang et al. [21] and Gavaises et al. [11] also observed that the bubble suction phenomenon at the end stage of injection from the nozzle in visual experiments, then when the needle valve left the seat again, a pressure wave strongly compressed the bubbles and the compressed bubbles were pushed out with the liquid. Besides, Battistoni et al. [22] have conducted a study, showing that residual gas can be ingested in the injector sac after the end-of-injection (EOI).

Due to the unavailability of the accurate transparent orifices, experimentally implementation and investigation are very difficult and only images of the spray are presented. While the numerical simulation can supply abundant data to make up for the inadequacies of experiments. And with ever increasing computational power, there is an incentive to use more complex models for primary atomization. Papadopoulos et al. [23] studied the flow inside and immediately downstream of a real-size multi-hole nozzle, during and after the injection event with a small air chamber coupled to the injector downstream of the nozzle exit based on the Volume Of Fluid (VOF) model, Reynolds Averaged Navier-Stokes (RANS) approach and cavitation models. However, Reynolds Averaged Navier Stokes (RANS) models cannot capture the transient jet structure [13,24], this limitation motivates the application of the LES model which directly resolves large scale eddies and models small eddies. The LES model is superior to the RANS for turbulence computations. The cavitation model of Schnerr-Saurer is widely used in the flow inside nozzles, which is compatible with all the turbulence models available in ANSYS FLUENT. The Volume Of Fluid (VOF) approach can model two or more immiscible fluids by solving a single set of momentum equations and tracking the volume fraction of each of the fluid throughout the domain. That is to say, VOF model can capture distinct liquid-gas interface boundaries. Thus, in this paper, a combination of the LES method, Schnerr-Saurer cavitation model and VOF phase-fraction model was applied to study the unsteady liquid jet flows at the early stage of injections coupled with the cavitating flow inside the orifice.

2. Methodology

2.1. Mathematical model

In this study, the described cases were simulated by using CFD software ANSYS Fluent 14.5. The LES method was used to solve the Navier–Stokes equations coupled with a mass transfer Schnerr-Saurer cavitation model and Volume Of Fluid (VOF) multi-phase flow model. The main features of the solver are given below.

The Dynamic Smagorinsky-Lilly Model (DSM) was applied to compute the sub-grid scale stress τ_{ij}, which was conceived and further developed by Germano et al. [25] and Lilly [26]. The equation can be described as follows:

$$\tau_{ij} = -2u_s\bar{S}_{ij} + \frac{1}{3}\tau_{kk}\delta_{ij} \tag{1}$$

Here, the value of the sub-grid eddy viscosity coefficient μ_s and the Smagorinsky model constant C_s can be computed via:

$$\mu_s = \rho C_s \Delta^2 |\bar{S}_{ij}| \tag{2}$$

$$C_s = \frac{L_{ij}M_{ij}}{M_{ij}M_{ij}} \tag{3}$$

Here, $\bar{S}_{ij}$ is the average rate of strain tensor, and the Δ represents the local grid scale. L_{ij} can be computed from the resolved large eddy field.

Schnerr-Sauer cavitation model was compatible with the LES method and is suitable for complex orifice flow conditions, which was developed by Schnerr and Sauer [27]. The model describes the growth of a single vapor bubble in a liquid based on the Rayleigh-Plesset equation, and uses the following two-phase continuity equations to derive the exact expression for the net mass transfer from liquid to vapor:

$$\frac{\partial}{\partial t}(\alpha\rho_v) + \nabla\cdot(\alpha\rho_v\vec{v}_v) = R_e - R_c \tag{4}$$

Here, ρ_v and $\vec{v}_v$ are the density and velocity of vapor phase, respectively. R_e and R_c are the source term of the rates of growth and collapse of bubbles, respectively. And α is the vapor volume fraction, which can be described as:

$$\alpha = \frac{n\cdot 4\pi R_B^3/3}{1 + n\cdot 4\pi R_B^3/3} \tag{5}$$

Here, n is the number density of bubbles per volume of liquid and R_B is the radius of bubbles. This expression associates the vapor volume fraction with the number of bubbles per volume of liquid. In this study, we took the value of n as 10^{13} [27]. The growth and collapse of bubbles are depicted by solving the following Rayleigh-Plesset equation [28]:

$$R_e = \frac{\rho_v\rho_l}{\rho}\alpha(1-\alpha)\frac{3}{R_B}\sqrt{\frac{2}{3}\frac{(p_V - p)}{\rho_l}} \quad p_V \geqslant p \tag{6}$$

$$R_c = \frac{\rho_v\rho_l}{\rho}\alpha(1-\alpha)\frac{3}{R_B}\sqrt{\frac{2}{3}\frac{(p - p_V)}{\rho_l}} \quad p_V \leqslant p \tag{7}$$

Here, p_V is the saturated vapor pressure of the diesel, and the value of p is usually taken as same as the cell center pressure.

The VOF model which relies on the fact that all phases involved are not interpenetrating each other was used to capture and evolve the interface. The tracking of the interfaces among the phases are accomplished by the solution of a continuity equation for the volume fraction [29]. For example, for the phase q, the equation form as follows:

$$\frac{1}{\rho_q}\left[\frac{\partial}{\partial t}(\alpha_q\rho_q) + \nabla\cdot(\alpha_q\rho_q\vec{v}_q) = S_{\alpha q} + \sum(\dot{m}_{pq} - \dot{m}_{qp})\right] \tag{8}$$

Here, $\dot{m}_{pq}$ is the mass transfer from phase p to phase q and $\dot{m}_{qp}$ is the mass transfer from phase q to phase p. $S_{\alpha q}$ is the source term for each phase which can be set based on the cavitation model. The diesel phase, diesel vapor phase, and air phases are taken into account in this study.

2.2. Numerical simulations

2.2.1. Physical model, boundary and initial conditions

In this paper, we simulated the process of diesel fuel injecting through a vertical single-hole nozzle. Both nozzle cavitating flow and near-nozzle jet spray are analyzed simultaneously in simulations. The geometry size and boundary conditions of the models are provided in Table 1. Under the assumption of cavitation formation at approximately constant pressure equal to saturation, the vapor density should be approximately constant. Moreover, the temperature used in this study is 293 K, which has a huge influence on fuel properties. The fuel properties are presented in Table 1.

In order to simplify the calculation, the upstream boundary of the computational domain was set at the top of sac and the needle was fixed. Considering that the asymmetry of the spray and the ligament or droplet formation was totally a three-dimensional phenomenon, two-dimensional and axisymmetric simulations may overlook essential dynamics. Thus, the hexahedral structured meshes of the full computational domain (360°) by ICEM were generated as shown in Fig. 1. There are 4 million cells in the computa-