International Communications in Heat and Mass Transfer 76 (2016) 108–116

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Experimental visualization and LES investigations on cloud cavitation shedding in a rectangular nozzle orifice☆

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

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

ARTICLE INFO

Available online 24 May 2016

Keywords:
Nozzle
avitation
Experimental visualization
LES
Shedding of cloud cavitation

ABSTRACT

Shedding of cloud cavitation in a nozzle orifice is a common phenomenon of cavitation instability which has a great influence on erosion damage and spray atomization process, but it hasn't been studied in details and the mechanism remains unclear. In this paper, visualization experiments and Large Eddy Simulation (LES) were carried out to study the process of periodic cloud cavitation shedding and re-entrant jet happened in a 2 mm-width nozzle with fuel of diesel. Numerical and experimental images provide a better insight into this physical mechanism. The results show that cavitation inception is formed at the edge of inlet and the recirculation zone which below the shear layer. Then the attached cavitation grows and shedding of cloud cavitation occurs due to the re-entrant jet located between cavity and wall. Moreover, the dynamic mechanism causing shedding of cloud cavitation is a combination of shedding vortex and pressure fluctuations. Cavitation shedding frequency can also be measured by experiment and numerical simulation. As a result, inception of cavitation, re-entrant jet and cavitation cloud shedding are accurately predicted by LES in accordance with the quantitative images observed in the experiment.

1. Introduction

The fuel atomization process and spray characteristics directly affect the mixture process of fuel and air. Then further the combustion process is extremely important in reducing fuel consumption and exhaust emissions in direct injection diesel engines. While the cavitating flow in diesel injector nozzles has a great influence on the fuel spray atomization, it is essential to make clear of this nozzle cavitating flow characteristics for better understanding of the spray primary atomization mechanisms.

Bergwerk [1] firstly reported the influence of cavitation on spray formation, and its connection with nozzle flow which was observed experimentally, and also found that a decrease in the discharge coefficient of the nozzle under cavitating conditions. However, the strong transient characteristics of the actual fuel injection process, small scales of the fuel injector nozzle and the lack of optical access make the experimental observation of the flow inside the injector nozzles in realistic conditions extremely hard. Therefore, most investigations have been carried out in a transparent scaled-up model at the aim of visualizing the cavity formation, growth and collapse process when occurrence of cavitation in diesel nozzles [2–3]. Payri et al. [4–5] reported that cavitation leads to an increase of the spray cone angle as well as flow outlet velocity, and measured the spray momentum in order to explain the effects of nozzle geometry on the cavitating flow. However, few attempts have been done to investigate the shedding of cloud cavitation in a diesel nozzle orifice.

The study of the unsteady cavitating flow which named cloud cavitation occurring on hydrofoils has reported that "re-entrant" jet is central to the process of cloud cavitation shedding which is a motion of a liquid jet beneath the attached cavity in the direction opposite to the main flow [6]. Therefore, for the diesel injector nozzle, studying the cavitating flow, especially the shedding of cloud cavitation in it will be crucially significant for better understanding of the motion of re-entrant jet and the increase of spray angle induced by this nozzle cavitating flow.

Recently, a number of studies [7–9] have tried to explore the mechanism of the cloud cavitation instability, both numerically and experimentally. It is typically thought that the re-entrant jet is created by the flow expanding in the closure region behind the cavity, impinging with the wall and establishing a local stagnation point [10]. Sato and Saito [11] performed visualization of periodic cloud shedding in large scale cylindrical orifices. Stanley and Barber [12] investigated the re-entrant jet mechanism for periodic cloud shedding cavitation experimentally, which showed a constant presence of a liquid sub-layer between the cloud cavitation and the nozzle wall, and it is also found that traveling pressure wave driven by the previous bubble collapse.

Therefore, with the rapid development of computing abilities and limitations of measurement techniques, a number of computational studies [13–16] on cavitation have been reported over the years. In most previous simulations, the Reynolds Averaged Navier–Stokes

☆ Communicated by W.J. Minkowycz.

* Corresponding author.
E-mail address: cyh5830739@163.com (Y. Chen).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2016.05.033

(RANS) approach like κ-ε models were used to treat turbulence. Actually, they do not address the influence of turbulent fluctuations on the onset and development of cavitation, and RANS models led cavitation to a steady-state solution. It seemed to be unable to predict the shedding of cloud cavitation and re-entrant jet. Accordingly, it is obvious that some more advanced turbulent models should be used to simulate this turbulent two-phase flow for capture of more details.

At present, Large Eddy Simulation (LES) has become a great alternative to advance the knowledge of the internal turbulent flow [17], as it is often more capable of reproducing large unsteady motion of the flow field. LES with the WALE SGS stress model was adopted to calculate the cavitation shedding and horse-shoe structures by Ji et al. [18].What's more , Ji also analyzed the pressure fluctuations induced by cloud cavitation around a hydrofoil. Sou et al. [19] proposed a new combination of Large Eddy Simulation (LES), Eulerian–Lagrangian Bubble Tracking Method (BTM), and the Rayleigh–Plesset (RP) equation to simulate an incipient cavitation in a scaled-up nozzle of fuel injector. The result showed that incipient cavitation flow can be predicted by LES using a fine grid. Egerer et al. [20] 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. Fuchs et al. [21] performed Large Eddy Simulations which not only are able to reproduce vortex cavitation, but also give further insight into the complex interaction between cavitation and turbulence.

In this study, we aim to investigate the shedding of cloud cavitation both in experimental and numerical methods. Additionally, induced pressure oscillations and turbulent vortex shedding were also analyzed with LES model to find out the relationship between cloud cavitation shedding and re-entrant jet. Given the importance of these phenomena on the subsequent mixing and spray processes, so the results hereby presented may be of interest for diesel nozzle orifice designers. To do so, a visual experimental system was set up to capture the images of the process of cloud cavitation in a 2 mm-wide rectangular orifice by a high speed CCD camera and supply the experimental data for verifications of numerical models.

2. Experimental setup

Aimed to further investigate the cloud cavitation shedding, a visualization experimental bench was set up. The schematic diagram of the visual experiment setup was shown in Fig. 1. Diesel was pumped by a gear pump to supply a working liquid to the rectangular nozzle. An electromagnetic flow meter and a pressure gauge were installed to measure the flow rate and static upstream injection pressure, respectively. About 0.8 m length of pipe before the nozzle inlet was used to reduce the upstream instabilities. It is noted that, nozzle flow inject into air so we keep the back pressure 0.1 MPa. The tested rectangular nozzle was made of transparent acrylic that has almost the same index of refraction as diesel fuel. The refractive index of acrylic is 1.49. Geometry of the rectangular nozzle was illustrated in Fig. 2. The width W and length L of the nozzle were about 2 and 4 mm, respectively. The nozzle had a W/L ratio of 2 and a nominally sharp entrance. This structure was representative for cavitating flow in fuel injectors, since the velocities and cavitation obtained are similar to the large-scaled fuel injector nozzle and it is easy for us to investigate cloud cavitation.

For verification of numerical simulation results, a high-speed CCD camera (MotionPro-TM10000, image size of 512 × 512 pixels at 10,000 fps) connected with a long distance microscope (Questar QM-1) was mounted on a tripod to capture the periodic behavior of cloud cavitation in the nozzle. It is clear in magnification and resolution when your specimen lies between 56 cm and1.5 m. The nozzle was illuminated by a high power LED light (99 LED bulbs) installed at the opposite side of the camera across the nozzle [22]. The cavitation number σ was defined as:

$$\sigma = \frac{p_0 - p_v}{0.5\rho v^2} \tag{1}$$

where P_0, P_v and v were the back pressure, the saturated vapor pressure and the mean liquid velocity in the nozzle, respectively. Here, we keep the back pressure around 0.1 MPa. The non-dimensional parameter can be well used for description of the cavitation conditions in nozzles.

3. Numerical models

The described case was simulated by using CFD software ANSYS Fluent 14.5. The spray cone angle and the breakup of fuel spray just out of the nozzle orifice are affected by the combination of turbulence and cavitation. The largest eddies are responsible for a good mixing and, consequently, for a good performance of the combustion reaction [23]. In addition, the shedding of cloud cavitation and re-entrant jet are complicated to simulate due to the highly transient behavior and the RANS turbulence model was not enough to capture transient cavitating flow. Thus, in this study, a combination of Large Eddy Simulation (treat the turbulent flow), a homogeneous multiphase mixture flow model (handling the interactions between the cavitation bubbles and the diesel flow field) and Schnerr–Saurer Cavitation model (capturing the formation of cavitation bubbles) was proposed to simulate the transient behavior of cloud cavitation.

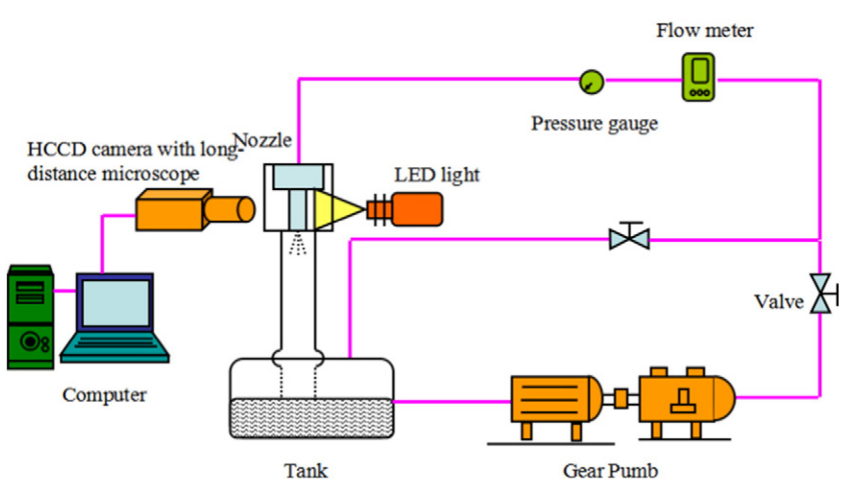

Fig. 1. Experimental setup.