Nuclear Engineering and Design 240 (2010) 3974–3983

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Flow structure of steam–water mixed spray

Toshiyuki Sanada*, Yuki Mitsuhashi, Hiroya Mizutani, Takayuki Saito

Department of Mechanical Engineering, Shizuoka University, 3-5-1 Johoku, Naka-ku, Hamamatsu 432-8561, Shizuoka, Japan

ARTICLE INFO

Article history:
Received 15 August 2009
Received in revised form 29 January 2010
Accepted 10 March 2010

ABSTRACT

In this study, the flow structure of a steam–water mixed spray is studied both numerically and experimentally. The velocity and pressure profiles of single-phase flow are calculated using numerical methods. On the basis of the calculated flow fields, the droplet behavior is predicted by a one-way interaction model. This numerical analysis reveals that the droplets are accelerated even after they are sprayed from the nozzle. Experimentally, the mixed spray is observed using an ultra-high-speed video camera, and the velocity field is measured by using the oarticle image velocimetry (PIV) technique. Along with this PIV velocity field measurement, the velocities and diameters of droplets are measured by phase Doppler anemometry. Furthermore, the mixing process of steam and water and the atomization process of a liquid film are observed using a transparent nozzle. High-speed photography observations reveal that the flow inside the nozzle is annular flow and that most of the liquid film is atomized at the nozzle throat and nozzle outlet. Finally, the optimum mixing method for steam and water is determined.

1. Introduction

A spray produced by atomization, which uses the disruptive action of a high-velocity gas upon a liquid stream, has a wide variety of applications. In these processes, the process of mixing of the gas with the liquid is important; therefore, several types of two-fluid nozzles, for example, internal or external mixing nozzles, have been proposed (Hede et al., 2008). Such two-fluid nozzles are mainly employed in chemical plants, fuel combustion equipment, and so on. As a result, many studies have focused on droplet diameter distribution because the surface area is the most important parameter. Further, the droplet speed is not so high.

Recently, it has been found that a high-speed steam–water mixed spray with a droplet speed of 200 m/s can be used for surface cleaning (Watanabe et al., 2009). It has also been found that an air–water mixed spray and a steam–water mixed spray have considerably different effects on a solid surface (Sanada et al., 2008). However, to the best of the authors' knowledge, a high-speed water spray containing steam instead of air has not yet been investigated in detail. Further, despite the fact that many studies have been conducted for investigating wet steam flows in a nozzle (Mccallum and Hunt, 1999; Geber, 2002; Jurski and Géhin, 2003; Geber and Kermani, 2004; Simpson and White, 2005), little information is available on the mixing process between steam and water, with the exception of a study on a steam injector (Cattadori et al., 1995; Narabayashi et al., 1997; Deberne et al., 1999; Dumaz et al., 2005). Therefore, in this study, we investigate the flow structure of a steam–water mixed spray, particularly the effect of the mixing process of steam and water on the spray shape. Further, we discuss the mixing process of steam and water, atomization of a liquid film, and droplet characteristics on the basis of experimental results obtained using high-speed photography, phase Doppler measurements, and numerical analysis. Finally, we determine an optimum mixing method for steam and water.

2. Numerical analysis

2.1. Numerical method and procedure

The flow field was calculated by numerical methods, using commercial CFD software ©FLUENT 6.2 and a mesh generator ©GAMBIT 2.3. Fig. 1 shows the three-dimensional (3D) mesh used for the calculation. Section A, shown in Fig. 1, is the grid used for calculation of the flow field inside the nozzle, and section B is the grid used for calculation of the jet flow under atmospheric pressure. First, single-phase flow fields were calculated. Both steam and air cases were solved. It should be noted that in this calculation, we assumed that the steam did not condense. The 3D flow fields were determined under steady conditions. A realizable-k–ε model was employed as a turbulent model. Boundary conditions such as no-slip on the wall and constant pressure and temperature at both the nozzle inlet and the atmospheric outlet were imposed. The inlet and atmospheric pressure and temperature were set to 0.2 MPa, 0.1 MPa, 393 K, and 288 K, respectively. In the calculation, first, a temporal solution was obtained by setting the inlet pressure to 0.11 MPa, i.e., a pressure difference of 0.01 MPa. Using this solu-

* Corresponding author.
E-mail address: ttsanad@ipc.shizuoka.ac.jp (T. Sanada).

0029-5493/$ – see front matter © 2010 Elsevier B.V. All rights reserved.
doi:10.1016/j.nucengdes.2010.03.020

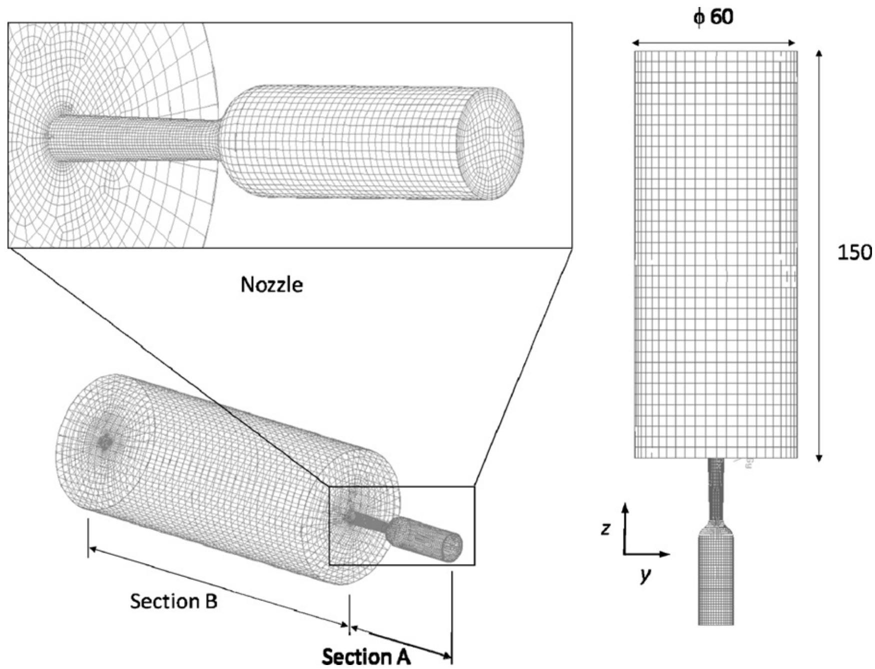

Fig. 1. Three-dimensional computational grid. Sections A and B are for the calculation of nozzle and jet, respectively.

tion as an initial condition, the next step solution was obtained at an inlet pressure of 0.12 MPa. This procedure was repeated up to an inlet pressure of 0.2 MPa. Finally, the convergence solution at a pressure difference of 0.1 MPa was obtained. After the calculation of the single-phase flow field, the droplet behavior was calculated using a one-way interaction model. Three different droplet diameters were tested—5 μm, 10 μm, and 20 μm. The dependence of the acceleration in the jet flow on the droplet diameter was calculated.

2.2. Numerical results and discussion

First, results of the numerical analysis are discussed. The profiles of velocity v, pressure p, and Mach number M on the center line in the steam and air cases are shown in Fig. 2. As shown in Fig. 2(a), in both cases, the velocity increases in the convergent region and still increases at the divergent region. However, it suddenly decreases before the nozzle outlet. The velocities at the nozzle outlet in air and steam cases are approximately 370 m/s and 470 m/s, respectively. As shown in Fig. 2(b), under the same pressure difference condition, the velocity of steam is approximately 100 m/s higher than that of air at the nozzle outlet. Further, Fig. 2(c) shows that there is no significant difference between the Mach number profiles in the steam and air cases; therefore, it is believed that the differences in the magnitudes of velocities of air and steam are a result of differences in their physical properties, particularly specific heat. This numerical analysis reveals that the velocities of steam are more accelerated than those of air, under the assumption that steam does not condense.

Next, the droplet behavior is discussed. Fig. 3(a) shows the velocity profiles of droplets of 5 μm, 10 μm, and 20 μm diameters. The velocity profile of steam is also plotted in the figure. These droplets were placed 50 mm upstream of the nozzle outlet with velocities of 0 m/s as the initial conditions. These droplets sharply accelerate in the nozzle and gradually decelerate after they attain the maximum velocity. The velocity profile of the 5 μm droplet is almost the same as that of steam. However, the 10 μm droplet attains the maximum velocity outside the nozzle, at $z = 30$ mm. This result shows that droplets of the order of 10 μm are still accelerated after they are sprayed from the nozzle. On the contrary, as shown in Fig. 3(b), the velocity profile of the droplet in air is almost the same as that of air itself. In the next section, a comparison of the numerical results with the experimental results is presented.

3. Experiments

3.1. Experimental apparatus and procedure

3.1.1. Experimental apparatus

Fig. 4 shows a schematic of the experimental apparatus. Steam was generated from water by electric heating and stored in a pressure tank. Steam and water were mixed before the nozzle inlet. The steam–water mixture was accelerated inside the nozzle and then sprayed. The steam pressure p in the gauge ranged from 0.05 MPa to 0.2 MPa. The water flow rate q ranged from 100 mL/min to 300 mL/min.

In this study, a transparent nozzle made of quartz was used to visualize the mixing process of steam and water and the flow structure in the nozzle atomization process. The flow structure in the nozzle was observed using a high-speed video camera (Photron, FASTCAM SA1.1) at 15,000 fps. In addition, the structure of the spray was captured by an ultra-high-speed video camera (Shimadzu, HPV-1) at 250,000 fps or 500,000 fps. A short arc strobe (Nissin, SA-200F) was used as a light source for the high-speed photography. This strobe is capable of emitting a very powerful light in a few hundred microseconds. A frosted glass was used to diffuse the light. In addition, in order to observe the atomization process inside the nozzle, a laser sheet (YVO4, 532 nm) was employed. The sheet thickness was controlled by using an optical system such as a beam expander or rod lens. The thickness of the laser sheet and the power were controlled to 0.6 mm and 300 mW, respectively. Fig. 5 shows the experimental setup of the optical system.

The droplet diameters and velocities were also measured. The spray velocity field was calculated by the particle image velocimetry (PIV) technique (FFT-based recursive cross-correlation method) applied to high-speed photographic image data. The interrogation region was downsized from 16 × 16 pixels to 8 × 8 pixels, and the overlap was 50%. The distribution of the droplet diameters and their velocities were also measured using a phase Doppler anemometer