International Journal of Heat and Mass Transfer 126 (2018) 831–842

Contents lists available at ScienceDirect

International Journal of Heat and Mass Transfer

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

# Numerical investigation on the effects of steam and water parameters on steam jet condensation through a double-hole nozzle

Lutao Wang, Xiaoyu Yue, Quanbin Zhao, Daotong Chong *, Junjie Yan

*State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, PR China*

ARTICLE INFO

*Article history:*
Received 9 January 2018
Received in revised form 8 May 2018
Accepted 16 May 2018

*Keywords:*
Condensation
Double-hole nozzle
Water temperature
Steam pressure
Thermal hydraulic parameters

ABSTRACT

Steam jet condensation through multi-hole nozzles in a pressure relief pool is important for the design and safe operation of a nuclear reactor system. In this study, stable steam jet condensation through a double-hole nozzle was investigated at different water temperatures and steam pressures by CFD method. Simulation results indicated that the shape of steam cavity changed from conical to ellipsoidal with an increase in water temperature and steam pressure, and steam jet length gradually increased. Meanwhile, the interaction between two steam cavities was enhanced and they even merged under certain conditions. Expansion and compression waves were found by analyzing the thermal hydraulic parameters along the hole centerline. Water temperature and steam pressure exerted different effects on the intensity of expansion/compression waves and the positions of maximum expansion/compression. Finally, thermal hydraulic parameters along the nozzle centerline were analyzed. Steam volume fraction, temperature, and velocity initially increased and then decreased as axial distance increased, which appeared as evident peaks under the present conditions. When water temperature and steam pressure increased, the peak values of steam volume fraction, temperature, and velocity gradually increased and their positions moved downstream.



## 1. Introduction

Steam jet condensation is a typical phenomenon of direct contact condensation (DCC) that has been widely investigated for its important industrial applications. Particularly, this phenomenon can be observed in a nuclear reactor pressure relief system. When the pressure in a nuclear reactor pressure vessel is higher than the safe threshold, steam with high pressure and temperature is injected into subcooled water pool to release pressure. Thus, the study of steam jet condensation has significant implication for the design and safe operation of a nuclear reactor system.

Various condensation patterns appear when steam condenses under different test conditions. Chan and Lee [1], Liang and Peter-Griffith [2], Cho et al. [3], and Wang et al. [4] presented their regime maps based on different water temperatures and steam mass fluxes. Cho et al. [3] observed six condensation patterns, namely, chugging, transient chugging, condensation oscillation, stable condensation, bubble condensation, and interfacial oscillation condensation, under a relatively wide range of operating conditions. When a reactor accident occurs, steam pressure is extremely high, such that steam will condense in a stable condensation region. Stable steam cavity will then form in the region. Many investigations of the steam cavity and flow field parameters of a stable steam jet have been conducted over the last three decades.

The shape of the steam cavity of a sonic jet through a single-hole nozzle was experimentally studied by Chun et al. [5], Kim et al. [6], and Wu et al. [7], in which conical, ellipsoidal, and divergent steam cavity shapes appear. They found that steam cavity shape depends on steam mass flux and water temperature. Maximum expansion ratio and steam jet length were used by Wu et al. [8] to describe the size of steam cavity. Certain correlations of steam jet length, which are related to water temperature and steam mass flux, have been proposed [5,6,9–11]. Kim et al. [6], Wu et al. [8], and Chong et al. [11] experimentally studied maximum expansion ratio and found that it increases with water temperature and steam mass flux. Moreover, the temperature distribution of steam jet condensation was measured by Kim et al. [6] and Wu et al. [8]. The results indicated that temperature distribution is influenced by steam and water parameters. However, the accurate measurement of the flow field parameters of steam cavity is difficult to achieve via experiment because of high-speed flow and equipment limit. The numerical method

* Corresponding author.
*E-mail addresses:* wanglutao@stu.xjtu.edu.cn (L. Wang), yuexiaoyu@stu.xjtu.edu.cn (X. Yue), zqb8031110@stu.xjtu.edu.cn (Q. Zhao), dtchong@mail.xjtu.edu.cn (D. Chong), yanjj@mail.xjtu.edu.cn (J. Yan).

https://doi.org/10.1016/j.ijheatmasstransfer.2018.05.090
0017-9310/© 2018 Published by Elsevier Ltd.

**Nomenclature**

| | |
|---|---|
| $A_{fg}$ | interfacial area per unit volume, /m |
| $d_g$ | steam bubble diameter, m |
| $d_e$ | nozzle diameter at the outlet, m |
| $d_0$ | bubble diameter at subcooling $\theta_0$, m |
| $d_1$ | bubble diameter at subcooling $\theta_1$, m |
| $h_f$ | heat transfer of liquid phase, W/m$^2$ K |
| $h_{fg}$ | latent heat, kJ/kg |
| $H_{fs}$ | saturation enthalpy of liquid phase, J/kg |
| $H_{gs}$ | saturation enthalpy of steam phase, J/kg |
| $m_{fg}$ | rate of mass transfer, kg/s |
| $N_{uf}$ | Nusselt number of liquid phase |
| $P_{in}$ | steam inlet pressure, kPa |
| $P_r$ | Prandtl number |
| $Q_f$ | total heat flux from liquid phase to the interface, W/m$^2$ |
| $Q_g$ | total heat flux from steam phase to the interface, W/m$^2$ |
| $q_f$ | sensible heat flux from liquid phase to the interface, W/m$^2$ |
| $Re_r$ | relative Reynolds number between steam and water |
| $T_f$ | liquid temperature, °C |
| $T_{sa}$ | saturation temperature, °C |
| $T_w$ | temperature of subcooled water, °C |
| X | axial length from nozzle inlet, m |
| Xe | axial length from hole exit, m |

*Greek letters*

| | |
|---|---|
| $\alpha_g$ | volume fraction of steam phase |
| $\theta$ | degree of liquid subcooling |

exhibits the advantage of providing more details inside steam cavity.

Several scholars have recently conducted investigations of steam jet condensation using numerical methods. Gulawani et al. [12,13] researched the flow pattern and heat transfer of the DCC process using a two-resistance model with computational fluid dynamics (CFD). Shah et al. [14] proposed a condensation model to simulate the steam jet condensation process. This condensation model was also used to research the DCC process in an ejector [15,16]. The steam jet condensation of sonic and supersonic nozzles was analyzed by Zhou et al. [17,18] by performing a steady CFD simulation at different steam inlet pressures and water temperatures. Li et al. [19] studied the DCC transient process in a tee junction at different temperatures.

In addition to single-hole nozzles, the steam jet condensation of multi-hole nozzles has also been studied by researchers. Cho et al. [20,21] experimentally concluded that the pressure oscillation characteristics of a multi-hole nozzle are influenced by water temperature and hole distribution modes. Park et al. [22,23] obtained the condensation regimes of steam jet condensation through a multi-hole nozzle with and without a bottom hole.

As stated earlier, scholars have given considerable attention to steam jet through a single-hole nozzle, but minimal attention has been devoted to steam jet through a multi-hole nozzle. To date, the mechanisms of steam jet through a multi-hole nozzle remain unclear. We obtained the radial distributions of thermal hydraulic parameters and the effect of pitch-to-hole diameter using CFD in our previous work [24]. The authors of [17] reported that the axial distribution of thermal hydraulic parameters can help identify the mechanism of steam jet, and steam and water parameters have significant effects on it. Accordingly, steam cavity and the axial distributions of thermal hydraulic parameters were investigated in this study, and the effects of water temperature and steam pressure were discussed in this paper. This research will contribute to the understanding of the DCC phenomenon and the safe design of relevant equipment.

## 2. Experimental apparatus

The correctness of the CFD model should be verified by comparing the simulated and experimental results. However, relative experiment data of steam jet through a double-hole nozzle are unavailable. Thus, an experiment on steam jet condensation through a double-hole nozzle was performed in this work for the sole purpose of qualitatively and quantitatively validating the correctness of the CFD model.

The experimental apparatus used in steam jet condensation, except the steam nozzle, was the same as that used in [8], as shown in Fig. 1(a). It consisted mainly of a steam generator, a surge tank, a water tank, and a high-speed video camera. Steam was generated by the steam generator and flowed through adiabatic pipes. Finally, steam was injected into subcooled water through a nozzle and condensed in the water tank. The water tank was rectangular and measured 3000 mm × 1000 mm × 1200 mm (length × width × height). The steam pressure of the nozzle inlet was measured with an absolute pressure transducer (range: 0–1.0 MPa; accuracy: 0.15% FS). Water temperature was measured using eight T-type thermocouples (accuracy: 1 °C). All the thermocouples were installed at the nozzle level, with two on the left wall, three on the front wall, and three on the back wall. A high-speed camera was attached to capture steam cavity photos with two underwater lamps as light sources. A T-type thermocouple was fixed on a mobile probe to measure temperature at different positions. Detailed experimental information is provided in [7,8].

Unlike the previous supersonic nozzle [8], the nozzle used in the experiment was a double-hole nozzle, as shown in Fig. 1(b). Two holes were distributed along the vertical direction (Y) and kept symmetric around the nozzle centerline. The detailed dimensions are shown in Fig. 1(b).

## 3. CFD simulation

### 3.1. Simulation zone and meshing solution

In this study, steam jet condensation in a stagnant water tank through a double-hole nozzle was investigated via CFD. The geometric model consisted of a cylinder and a double-hole nozzle, as shown in Fig. 2. The diameter of the nozzle inlet was 50 mm. The diameter of both holes was 8 mm, and the center distance was 12 mm. Two holes were distributed along the vertical direction and kept symmetric around the nozzle centerline. The diameter and length of the cylinder fluid zone were 300 mm and 680 mm, respectively. The cylinder fluid zone can be used to simulate the actual process of stable steam jet because the outer surface is far from the nozzle exit. The treatment was similar to the studies of Shah et al. [14] and Zhou et al. [17,18]. In the simulation work, a structured grid was selected to mesh the geometric model. The grid distribution is shown in Fig. 3. Grid density was high near the nozzle exit and between the two holes in the complex flow field, but relatively low at the downstream section for a smooth change in the flow field parameters.