International Communications in Heat and Mass Transfer 83 (2017) 48–54

Contents lists available at ScienceDirect

International Communications in Heat and Mass Transfer

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

Numerical investigation on steam jet submerged in subcooled water under different ambient pressures

Lun Zhou [a], Jiping Liu [a,*], Weixiong Chen [b], Daotong Chong [b], Junjie Yan [b]

[a] MOE Key Laboratory of Thermal Fluid Science and Engineering, Xi'an Jiaotong University, Xi'an 710049, PR China
[b] State Key Laboratory of Multiphase Flow in Power Engineering, Xi'an Jiaotong University, Xi'an 710049, PR China

ARTICLE INFO

Available online 9 March 2017

Keywords:
Condensation jet
Ambient pressure
Flow pattern
Thermal hydraulic

ABSTRACT

This paper aims to investigate the effect of ambient pressure on the thermal hydraulic behavior of stable steam jet during the process of direct contact condensation (DCC). Three-dimensional steady CFD simulation was conducted. A thermal equilibrium phase change model was inserted as a user defined function (UDF) to simulate the process of steam jet condensed into subcooled water. The shape of steam plume and axial temperature distribution were compared between experimental and numerical results, and good agreements were obtained. The steam plume shapes under different ambient pressures were obtained. The transformation of flow pattern from annular flow to bubble flow along axial direction was observed from the cross-sectional slices at different axial locations. Then the axial parameters such as velocity, temperature and static pressure under different ambient pressures were investigated. The existence of expansion and compression wave was verified due to the existence of the fluctuation of axial temperature. Besides, the average heat transfer coefficient ranges from 0.97 to 1.08 MW/m^2 K when ambient pressure ranges from 80 to 200 kPa.

1. Introduction

Due to the advantage of high capacity of heat and mass transfer, direct contact condensation (DCC) is widely used in industrial applications, such as the safety depressurization system, negative water-supply system in advanced pressurized-water reactor (PWR), feed water system of boiling water reactor (BWR), mixing-type heat exchanger and two-phase flow steam injector, etc.

When steam was injected into quiescent subcooled water, condensation modes including chugging, bubbling and jet were reported by Liang and Griffith [1]. Based on the flow conditions of steam at the nozzle exit, steam jet can be divided into subsonic jet, sonic jet and supersonic jet [2]. According to the condensation pattern, it can be divided into stable jet and unsteady jet [3]. Regime maps for DCC were studied by Chan and Lee [3], Cho et al. [4], Petrovic et al. [5], Wu et al. [6]. When stable steam jet occurs, there exists a stable steam region, called as steam plume. As for sonic steam jet, conical and ellipsoidal steam plume shapes were observed by Kostyuk [7], Del Tin et al. [8], Chun et al. [9], Kim et al. [10]. As for supersonic steam jet, there exist six types of steam plume shapes according to Wu's work [2]. The flow field of steam jet was measured by Kim et al. [10] and Wu et al. [11,12]. As far as numerical work is concerned, related work is relatively limited. As for chugging flow, numerical work was conducted by some researchers [13–15]. When it comes to stable steam jet, the work was conducted by Gulawani et al. [16], Shah et al. [17] and Zhou et al. [18]. Steam jet pump was also computationally investigated by Shah et al. [19–21]. With respect to stable steam jet submerged in quiescent water, the internal detail in steam plume is not clear till now. Additionally, related studies were all performed under 1.0 bar pressure whether it was from experimental or numerical work. The investigation about the effect of ambient pressure on steam jet submerged into quiescent water is absent. However, the ambient pressure may be not 1.0 bar pressure in actual industrial occasions. Consequently, this paper aims to investigate the effect of ambient pressure on steam jet, and this will be helpful to the understanding of DCC and safe operation of relevant DCC equipment.

2. Physical model & numerical method

2.1. Physical model

This paper concentrates on stable steam jet submerged in a water pool from a sonic nozzle. Related experimental work has been conducted by Wu et al. [11]. The detail dimensions of steam nozzle are shown in Fig. 1, whose unit is mm. In this study, the simulations were performed on a cylinder domain, which is shown in Fig. 2. Steam was accelerated through a converging nozzle and was injected into a water pool, then DCC occurs. The diameter of cylinder fluid zone and steam inlet is 300 and 50 mm, respectively. The length of fluid zone is 800 mm. Because

* Corresponding author.
E-mail addresses: zl3112009010@stu.xjtu.edu.cn (L. Zhou), liujp@mail.xjtu.edu.cn (J. Liu).

http://dx.doi.org/10.1016/j.icheatmasstransfer.2016.11.004
0735-1933/

Nomenclature

A_i	area of steam plume, m^2
d_e	diameter of nozzle exit, m
d_{ex}	maximum diameter of steam plume, m
G_e	steam mass flux, kg/m^2 s
H	average heat transfer coefficient, MW/m^2 K
h_{fg}	the latent heat of steam, kJ/kg
l/d	dimensionless penetration length
m_s	steam mass flow rate, kg/s
P_{in}	pressure at the inlet of steam nozzle, kPa
P_b	ambient pressure, kPa
P_s	static pressure, kPa
T	temperature, °C
T_w	temperature of subcooled water in the pool, °C
T_{in}	temperature of steam at the inlet of steam nozzle, °C
V_g	velocity of steam along axial direction, m/s
X	axial distance from nozzle inlet, m
X_e	axial distance from nozzle exit to the cross-sectional slice, mm

the distance between the outer surface of fluid zone and the nozzle exit is relatively far, the cylinder fluid zone can be used to simulate the whole flow field in the experiment [11] approximately. ICEM software was employed to generate structure mesh for calculation domain. The density of mesh near the nozzle exit was relatively large. The mesh solution can be seen from Fig. 2.

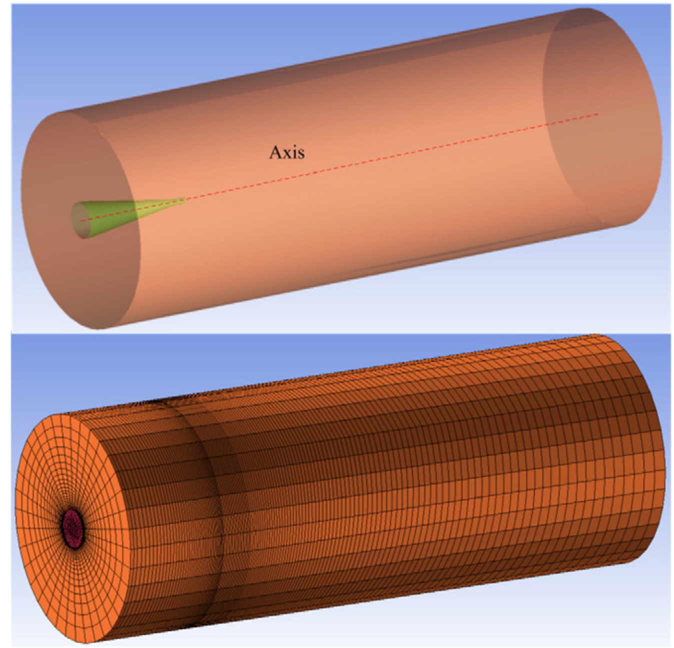

Fig. 2. Sketch of simulation zone and grid solution [18].

2.2. Numerical method

2.2.1. Simulation set-up

Three-dimensional steady simulation was carried out to simulate the process of stable steam jet in Ansys. Realizable $k-\varepsilon$ turbulent model was used. Simple algorithm was employed for pressure-velocity coupling, and implicit scheme was used in time discretization. Standard wall function was adopted. Gravity effect was concerned. Steam was treated as compressible gas using ideal gas law to calculate the density. The other properties of steam, such as heat capacity, conductivity and viscosity were set to be constant during the simulation. The symmetric model was used for drag formulation between the two phases. First order upwind scheme was used for spatial discretization for the sake of convergence during the simulation. Eulerian multiphase model was chosen as the multiphase model, and the details of the models mentioned above can be referred to the manual of Fluent [22].

2.2.2. Condensation model

A thermal equilibrium phase change model was inserted as a UDF (user defined function) to simulate the condensation process of steam jet. It is based on three key parameters: interfacial area, interfacial heat transfer coefficients and interfacial mass transfer [15,17–21]. Specification of volumetric heat transfer coefficient requires an estimate of the interfacial area per unit volume A_{fg}. For the flow of spherical bubbles of diameter d_g and with steam void fraction α_g in a liquid, the interfacial area per unit volume is estimated by:

$$A_{fg}=\frac{6\alpha}{d_g} \tag{1}$$

The mean bubble diameter is modeled as a linear function of local liquid subcooling ($T_s - T_f$):

$$d_g=\frac{d_1(\theta-\theta_0)+d_0(\theta_1-\theta)}{\theta_1-\theta_0} \tag{2}$$

where d_0 and d_1 are the bubble diameters at the reference liquid subcoolings θ_0 and θ_1. The reference subcoolings and bubble diameters recommended are $d_0 = 1.5\times10^{-4}$ m at $\theta_0=13.5$ K and $d_1 = 1.5\times10^{-3}$ m at $\theta_1=0$ K. The heat transfer coefficients between fluids are required to perform the heat and mass transfer calculations. The heat transfer between the two fluids was modeled in two steps: from vapor to the interface and from the interface to liquid characterized by two heat transfer coefficients for both sides of the interface. The liquid phase heat transfer coefficient h_f is related to the Nu_f by:

$$h_f=\frac{k_f Nu_f}{d_g} \tag{3}$$

The Nusselt number for liquid phase is calculated using the correlation as given below:

$$Nu_f=\begin{Bmatrix} 2.0+0.6{Re_r}^{0.5}Pr^{0.33} & 0<=Re_r<776.06 \\ 2.0+0.27{Re_r}^{0.62}Pr^{0.33} & Re_r>776.06 \end{Bmatrix} \tag{4}$$

The volumetric heat transfer coefficient for liquid phase is:

$$H_f=h_f A_{fg} \tag{5}$$

Zero thermal resistance was assumed from steam to interface, consequently, the temperature difference between the interface and

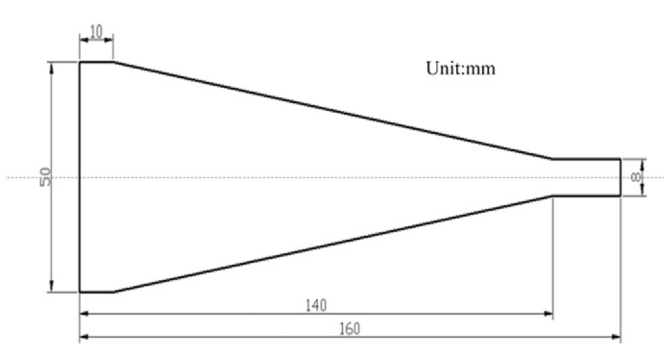

Fig. 1. Detailed dimensions of sonic nozzle [11].