Progress in Nuclear Energy 85 (2015) 729–746

Contents lists available at ScienceDirect

Progress in Nuclear Energy

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

# CFD based approach for modeling steam–water direct contact condensation in subcooled water flow in a tee junction

S.Q. Li [a], P. Wang [b], T. Lu [a], *

[a] College of Mechanical and Electrical Engineering, Beijing University of Chemical Technology, Beijing, 100029, China
[b] School of Energy and Power Engineering, Dalian University of Technology, Dalian, 116024, China

A R T I C L E I N F O

*Article history:*
Received 28 May 2015
Received in revised form
28 August 2015
Accepted 14 September 2015
Available online 22 September 2015

*Keywords:*
Direct contact condensation
Numerical simulation
Tee junction
Thermal cycling
Water hammer

A B S T R A C T

The phenomenon of direct contact condensation (DCC) is encountered in several components of a nuclear power plant in case of some transient and accident conditions. Studies on steam jet discharged directly into a stagnant water pool have been studied extensively; however, it is sparse for direct contact condensation of steam in subcooled water in pipes. The present paper aims to investigate the low mass flux saturated steam discharged directly into subcooled water flow in a tee junction. The Volume of Fluid (VOF) two-phase flow model and Large Eddy Simulation (LES) turbulent flow model of FLUENT have been used. Also, a thermal equilibrium model considering heat and mass transfer was introduced to model steam condensation using user-defined function (UDF). The simulation results show that, for steam mass flux of 10 kg/m$^2$ s, large chugging, small chugging and bubbling are obtained at water temperature of 303.15 K, 343.15 K and 363.16 K, respectively. In addition, it was found that, in chugging, there exist temperature and pressure oscillations at the location where the vertical branch pipe and the horizontal main pipe intersect. The phenomena of temperature and pressure oscillations at this location are called thermal cycling and water hammer, respectively. Moreover, the pressure oscillation frequency increases and the spike pressure decreases with increasing of subcooled water temperature in chugging. The Computational Fluid Dynamics (CFD) simulation results were qualitatively validated against the test results. It would be possible to improve the accuracy of the test results by employing multiple cameras.

## 1. Introduction

The phenomenon of direct contact condensation (DCC) where saturated steam condenses on the subcooled water interface concerns the safety of nuclear reactors. DCC can appear in the nuclear reactor emergency core cooling system and the thermal loops where steam and subcooled water can mix. When DCC occurs, quick and dangerous transients can cause thermal and pressure oscillation, which leads to undermine the structural integrity of related equipment and piping. Thus, it is necessary to carry out research on DCC mechanism for the sake of design and safety operation of relevant equipment and piping.

In the past decades, a lot of studies on DCC of steam jet discharged into a stagnant water pool have been studied extensively, experimentally and theoretically. Kerney et al. (Kerney et al., 1972) studied the length of the turbulent vapor cavity formed by a steam jet discharging into a subcooled liquid water bath and developed a correlation to predict the steam plume length. Chan and Lee (Chan and Lee, 1982) and Lahey and Moody (Lahey and Moody, 1993) experimentally studied steam injected into a pool of subcooled water and presented similar regime maps for DCC, which depend on the liquid temperature and the steam mass flux. Nariai (1986) experimentally and theoretically carried out pressure and fluid oscillations induced by DCC of steam flow with cold water at Emergency Core Cooling Water Injection in water-cooled nuclear power reactors. Furthermore, they presented classifications and mechanisms of these oscillations. Aya et al. (Aya and Narini, 1991) investigated the heat transfer coefficient at direct contact condensation of cold water and steam and reported that the heat transfer coefficient of chugging is larger than that of condensation oscillation. Kim et al. (Kim et al., 1997) experimentally studied direct contact condensation of steam jets injected into the subcooled water. The effect of steam mass flux and liquid subcooling on steam plume shape, steam plume length and the average heat transfer coefficient were discussed. Youn et al. (Youn et al., 2003) studied the direct contact condensation of steam at low mass flux

---

* Corresponding author.
*E-mail address:* likesurge@sina.com (T. Lu).

http://dx.doi.org/10.1016/j.pnucene.2015.09.007
0149-1970/

**Nomenclature**

| | |
|---|---|
| $A_{fg}$ | interfacial area per unit volume ($m^{-1}$) |
| $C_p$ | specific heat of the liquid ($J\ kg\ K^{-1}$) |
| $d_e$ | steam pipe diameter (m) |
| $d_g$ | bubble diameter (m) |
| $d_0$ | bubble diameter at liquid subcooling $\theta_0$ (m) |
| $d_1$ | bubble diameter at liquid subcooling $\theta_1$ (m) |
| $G_e$ | steam mass flux ($kg\ m^{-2}\ s^{-1}$) |
| $H_f$ | volumetric interfacial heat transfer coefficient of liquid phase ($W\ m^{-3}\ K^{-1}$) |
| $H_{fs}$ | saturation enthalpies of liquid phase ($J\ kg^{-1}$) |
| $H_g$ | volumetric interfacial heat transfer coefficient of vapor phase ($W\ m^{-3}\ K^{-1}$) |
| $H_{gs}$ | saturation enthalpies of vapor phase ($J\ kg^{-1}$) |
| $h_f$ | heat transfer coefficient of the liquid phase ($W\ m^{-2}\ K^{-1}$) |
| $h_g$ | heat transfer coefficient of the vapor phase ($W\ m^{-2}\ K^{-1}$) |
| $k_f$ | thermal conductivity of liquid phase ($W\ m^{-1}\ K^{-1}$) |
| $\dot{m}_{fg}$ | rate of mass transfer ($kg\ s^{-1}$) |
| $Nu_f$ | Nusselt number of liquid phase |
| $P_r$ | Prandtl number of liquid phase |
| $Q_f$ | total heat flux from liquid phase to the interface ($W\ m^{-2}$) |
| $Q_g$ | total heat flux from vapor phase to the interface ($W\ m^{-2}$) |
| $q_f$ | sensible heat flux from liquid phase to the interface ($W\ m^{-2}$) |
| $q_g$ | sensible heat flux from vapor phase to the interface ($W\ m^{-2}$) |
| $Re$ | relative Reynolds number |
| $Re_e$ | Reynolds number of vapor phase at the pipe inlet |
| $T_f$ | liquid temperature (K) |
| $T_g$ | vapor temperature (K) |
| $T_s$ | saturation temperature (K) |
| $U_f$ | velocity of liquid phase ($m\ s^{-1}$) |
| $U_g$ | velocity of vapor phase ($m\ s^{-1}$) |
| $\alpha_g$ | volume fraction of vapor phase |
| $\theta$ | degree of liquid subcooling (K) |
| $\mu_f$ | viscosity of liquid phase ($kg\ m^{-1}\ s^{-1}$) |
| $\rho_f$ | density of liquid ($kg\ m^{-3}$) |

(in the chugging region) and the pressure oscillation induced by condensation. A critical value of steam mass flux was found where the pressure pulse generation rate increased suddenly.

Recently, Puustinen et al. (Puustinen et al., 2013) performed experimental studies for the formation and condensation of steam bubbles at the blowdown pipe by using the facility of POOLEX, which was first test facility constructed for BWR containment studies. Qiu et al. (Qiu et al., 2014) experimentally studied the pressure oscillation of sonic steam jet in a pool and developed correlations to predict the dimensionless R.M.S (root mean square) amplitude of pressure oscillation. Rassame et al. (Rassame et al., 2015) carried out experiment to understand the void behavior in the pressure suppression chamber during the blowdown period of a LOCA. Three distinct phases were observed, namely an initial phase, a quasi-steady, and a chugging phase.

Although a lot of studies on DCC of steam jet discharged into a stagnant water pool have been previously extensively studied, little information is available on DCC of steam jets in water flow in pipes. Akimoto et al. (Akimoto et al., 1983) developed a two-phase flow model to formulate the condensation and mixing processes in the injection region used in case of violent condensation takes place in cold legs because of direct contact of steam with water. Xu et al. (Xu et al., 2013) experimentally studied the direct contact condensation of stable steam jet in water flow in a vertical pipe. The plume shape, plume length, temperature distribution, average heat transfer coefficient, and average Nusselt number were investigated. In 2015, Xu and Guo. (Xu and Guo, in press) carried out another experiment to investigate the flow and geometry characteristics of DCC of steam jet in crossflow of water in a vertical pipe. The results suggested that the condensation regime is not only affected by steam mass flux and water temperature, but also affected significantly by Reynolds number of water flow.

In the field of nuclear safety analysis, Computational Fluid Dynamics (CFD) has become an increasingly applicable tool for thermalhydraulic investigations (Bestion, 2012). However, the numerical simulations of DCC phenomena are rather sparse compared with that of experimentation methods. Furthermore, to our knowledge, there is no published work on direct contact condensation of low mass flux steam jet in subcooled water flow in a tee junction so far. Gulawani et al. (Gulawani et al., 2006) carried out three-dimensional CFD simulations to study the phenomena of direct-contact condensation using the Thermal Phase Change model of commercial CFD code CFX. Shah et al. (Shah et al., 2010) performed CFD simulation of DCC of supersonic steam in subcooled water using the commercial CFD software Fluent 6.3, and the simulation results were in fairly good agreement with the published experimental data. Jeon et al. (Jeon et al., 2011) carried out CFD simulation of steam bubble condensing in water using the volume of fluid (VOF) model in the FLUENT code. The bubble condensation was modeled by the user-defined function (UDF) and the simulation results were compared with experimental data. Tanskanen et al. (Tanskanen et al., 2014) utilized the NEPTUNE_CFD code to simulate chugging phenomena using Eulerian model with heat and mass transfer. Patel et al. (Patel et al., 2014) conducted CFD simulations of DCC at very low steam mass flux using NEPTUNE_CFD and OpenFOAM. The present authors' group has also performed investigations of CFD simulations on DCC of steam in water pool using the UDF to model steam condensation. Numerical simulation results were qualitatively compared with the published experimental data, and fairly good agreement was found between the two (Li et al., 2015).

As for the CFD simulation of DCC, especially for chugging region, it is important to capture the fluctuating details. While it has been commonly observed that Reynolds averaged Navier—Stokes (RANS) turbulence models are unable to capture the fluctuating flow details, large-eddy simulation (LES) and similar scale-resolving turbulence models have been found suitable (Timperi, 2014). As for simulation of two-phase flow, VOF method could be used to solve the advection equation of the volume fraction and predict the interface accurately. Because the VOF method has the advantage of superior volume-conservation compared to any other fixed grid interface or volume-tracking methodology (Muñoz-Esparza et al., 2012). Thus, it is considerable to model the phenomena of steam injection into subcooled water by VOF and LES method.

This investigation aims to perform the CFD simulation of DCC of steam in water flow in a Tee junction using UDF of condensation model considering heat and mass transfer. Accuracy of CFD simulations was qualitatively assessed by comparison with a steam—-water two phase flow experiment. The condensation model, the two resistance model, has been developed for the direct-contact