International Journal of Thermal Sciences 104 (2016) 194–207

Contents lists available at ScienceDirect

International Journal of Thermal Sciences

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

Poly-disperse simulation of condensing steam-water flow inside a large vertical pipe

Yixiang Liao*, Dirk Lucas

Helmholtz – Zentrum Dresden-Rossendorf, Institute of Fluid Dynamics, Bautzner Landstraße 400, 01328 Dresden, Germany

ARTICLE INFO

Article history:
Received 30 June 2015
Received in revised form
14 January 2016
Accepted 17 January 2016
Available online 11 February 2016

Keywords:
Poly-disperse
CFD simulation
Condensation
Steam-water
Coalescence and breakup

ABSTRACT

The condensation of saturated steam bubbles in sub-cooled water inside a vertical pipe was studied by poly-disperse CFD simulations. Six test cases with varied pressure, liquid sub-cooling and diameter of the gas injection orifices were simulated. Baseline closures presented for non-drag forces in previous work were found to be reliable also in non-isothermal cases. The effect of bubble coalescence and breakup is over-weighting in the region close to steam injection in case of small orifice diameter. With the increase of orifice diameter, breakup becomes dominant in determining bubble size change. The effect of interphase heat transfer coefficient correlations was investigated. The widespread Ranz–Marshall correlation was found to under-estimate the condensation rate, especially at high pressure levels. In contrast, satisfying agreement with the experimental data was obtained by the Tomiyama correlation.

1. Introduction

Phenomena of gas–liquid flows, which are actually omnipresent in our daily life for example as you boil a pot of water, have a significant meaning for industrial applications and are for this reason also in the focus of scientific investigations. For safety and economic concerns, knowledge of such flows is of ultimate importance. Nowadays owing to the ongoing increasing of available computational power, computer-aided simulation of CFD (Computational Fluid Dynamics) has been becoming an indispensable analysis tool also for two-phase flow phenomena. For example, the Eulerian Two-Fluid Model (TFM) has been used widely and successfully for the simulation of isothermal gas–liquid flows such as bubble plume, bubble column, vertical pipe flows, and phase separation in a T-junction. Recently, non-isothermal flows have also been simulated, e.g. condensation [1,2], wall boiling [3,4] as well as flashing and cavitation [5–10]. Nevertheless, in the application of TFM a great effort is still needed for the development of closure models especially for the non-isothermal cases. In these cases interphase mass, momentum and energy exchanges take place simultaneously and are strongly coupled with each other. There is still no broad consensus among various research groups even for isothermal cases, which is on the other hand important for comparison study and further improvement work.

In previous works, e.g. Refs. [11–14], a specification of closures for bubble forces, bubble-induced turbulence as well as bubble coalescence and breakup, was acknowledged as a baseline for further development. Using the well-defined set of closures according to the baseline definition, results with acceptable accuracy were achieved for both mono- and poly-disperse isothermal air–-water flows without the need of any adjustments. The promising results encourage us to validate them for other conditions. In the current work, the baseline closure models are extended to steam-water flows and completed with interphase mass and heat transfer. Due to limited space, only closures related to interphase mass and heat transfer are introduced here. The others remain the same as those in the isothermal cases since so far no concrete information regarding the effect of phase change is available. For more details on these closures the reader is referred to [14].

The reminder of the paper is organized as follows. After a short description of the investigated test cases in Section 2, detailed information about the simulation setup and mathematical models is given in Section 3. Section 4 describes mesh statistics and mesh dependency study while major results and comparisons with experimental data are presented in Section 5. Finally, a short conclusion with an outlook for future work ends the paper.

* Corresponding author. Tel.: +49 (0) 3512602389.
E-mail address: Y.Liao@hzdr.de (Y. Liao).

http://dx.doi.org/10.1016/j.ijthermalsci.2016.01.016
1290-0729/

2. Experiment and test cases

The condensation of saturated steam in sub-cooled water inside a large vertical pipe is simulated with the commercial CFD code ANSYS CFX 14.5. Experimental investigations were carried out previously at the TOPFLOW test facility and introduced in Refs. [15–17]. The test section consists of a pipe with an inner diameter of 195.3 mm and a length of about 8 m. Steam is injected via orifices in the pipe wall. There are injection chambers with a ring of 72 equally distributed orifices of 1 mm diameter and chambers with a ring of 32 equally distributed orifices of 4 mm diameter. They assure a rotation-symmetric gas injection. Using the different chambers allows the variation of the initial bubble size distribution. There are altogether around 100 test cases covering several injection pressures, superficial velocities (J_l, J_g), degree of initial sub-cooling (ΔT_{in}) and diameter of the gas injection orifices ($D_{orifice}$). Six of them are investigated numerically in the current work. The details of these cases provided by the experiment are summarized in Table 1. They have the same superficial velocities, but differ in operational pressure, initial sub-cooling and orifice diameter. The initial sub-cooling ΔT_{in} is measured before steam injection. The impact of these parameters on the flow behaviour will be analyzed below. Particle Reynolds number of the six test cases lies in the range of $0.5–3.0 \times 10^4$.

Data acquired with a wire-mesh sensor and a lance of thermocouples, are present for different levels. The height of each level relative to the steam injection elevation is given in Table 2. The height positions are distinguished by the orifice diameter, one with $D_{orifice} = 1.0$ mm and the other $D_{orifice} = 4.0$ mm.

The experimental results show that upward co-current flow prevails in the whole pipe. Nevertheless, vortex exists in the region below Level A/B, and it disappeared before it reached Level A/B. With consideration of numerical stability, measurements at Level A or Level B were used as inlet conditions in the simulation of cases with $D_{orifice} = 1.0$ mm and $D_{orifice} = 4.0$ mm, respectively. Furthermore, the inlet velocity was approximated with the vertical component while the radial one was ignored. The approximation has little influence on the simulation results, since the vertical component is absolutely dominant and the radial migration of bubbles can be rapidly adjusted owing to the effect of bubble forces.

3. Model setup

3.1. Fundamental transport equations

In the current work, the Eulerian two-fluid model is applied, where water is treated as continuous phase. It exchanges mass, momentum and energy with steam, which is dispersed in the form of discrete bubbles. The steam is divided further into two numerical phases with consideration of sign change in the lift force

Table 1
Selected test cases.

Case no.	J_l [m/s]	J_g [m/s]	ΔT_{in} [K]	$D_{orifice}$ [mm]
p = 10 bar				
1	1.017	0.219	3.9	1.0
2	1.017	0.219	5.0	1.0
p = 20 bar				
3	1.017	0.219	3.7	1.0
4	1.017	0.219	6.0	1.0
5	1.017	0.219	6.0	4.0
p = 40 bar				
6	1.017	0.219	5.0	1.0

Table 2
Heights relative to steam injection position.

$D_{orifice} = 1.0$ mm		$D_{orifice} = 4.0$ mm	
Level	Height [m]	Level	Height [m]
A (inlet)	0.221	B (inlet)	0.278
C	0.335	E	0.551
F	0.608	H	1.495
I	1.552	K	2.538
L	2.595	N	4.474
O	4.531		

coefficient. Small bubbles have a positive coefficient while large bubbles have a negative [18]. The dependence of the critical bubble size for sign change on the pressure in steam-water systems is presented in Ref. [19].

The ensemble-averaged mass, momentum and energy conservation equations for individual phases are given below. The steam is assumed to have always the saturation temperature corresponding to local absolute pressure, namely, $T_g = T_{sat}$. In addition, the temperature distribution inside the steam bubble is uniform. The rate of heat transfer from the interface to the adjacent fluid is proportional to the temperature difference ΔT between the interface and the fluid. That means that on the liquid side ΔT is equal to $T_i - T_l$, while on the steam side $T_i - T_g$, where T_i represents the interface temperature. Since the interface is assumed to have the saturation temperature too, the heat flux from the interface to the steam is zero. Therefore, no energy equation is necessary for it. On the other hand, the enhancement of water temperature due to the heat input from the interface or the condensation is described by the total energy equation Eq. (3) below.

Continuous phase (Water):

$$\frac{\partial}{\partial t}(\alpha_l\rho_l) + \nabla\cdot(\alpha_l\rho_l\vec{U}_l) = \sum_{k=1}^{2}\Gamma_{lg,k}, \tag{1}$$

$$\frac{\partial}{\partial t}(\alpha_l\rho_l\vec{U}_l) + \nabla\cdot(\alpha_l\rho_l\vec{U}_l\vec{U}_l) = -\alpha_l\nabla p + \nabla\cdot(\alpha_l\vec{\tau}_l) + \alpha_l\rho_l\vec{g} + \sum_{k=1}^{2}\vec{F}_{lg,k} + \sum_{k=1}^{2}\Gamma_{lg,k}\vec{U}_{g,k}, \tag{2}$$

$$\frac{\partial}{\partial t}\left(\alpha_l\rho_l\left(H_{tot,l} - \frac{p}{\rho_l}\right)\right) + \nabla\cdot(\alpha_l\rho_l\vec{U}_l H_{tot,l}) = \nabla\cdot\left(\alpha_l\lambda_{eff}\nabla T_l\right) + \dot{q}_l + \sum_{k=1}^{2}\Gamma_{lg,k}H_{tot,l,sat}. \tag{3}$$

Dispersed phase (Steam):

$$\frac{\partial}{\partial t}\left(\alpha_{g,k}\rho_{g,k}\right) + \nabla\cdot\left(\alpha_{g,k}\rho_{g,k}\vec{U}_{g,k}\right) = -\Gamma_{lg,k}, \tag{4}$$

$$\frac{\partial}{\partial t}\left(\alpha_{g,k}\rho_{g,k}\vec{U}_{g,k}\right) + \nabla\cdot\left(\alpha_{g,k}\rho_{g,k}\vec{U}_{g,k}\vec{U}_{g,k}\right) = -\alpha_{g,k}\nabla p + \nabla\cdot\left(\alpha_{g,k}\vec{\tau}_{g,k}\right) + \alpha_{g,k}\rho_{g,k}\vec{g} - \vec{F}_{lg,k} - \Gamma_{lg,k}\vec{U}_{g,k}, \tag{5}$$

where the subscript k = 1 or 2 indicates two dispersed phases representing small and large steam bubbles, respectively. $\Gamma_{lg,k}$ and $\vec{F}_{lg,k}$, $\dot{q}_l$ represent the volumetric mass and momentum transfer rate across the interface, respectively. Interfacial forces such as drag, lift, turbulent dispersion, virtual mass and wall lubrication forces are considered in the current work. A detailed description of