Chinese Journal of Chemical Engineering 23 (2015) 1460–1468

Contents lists available at ScienceDirect

Chinese Journal of Chemical Engineering

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

Fluid Dynamics and Transport Phenomena

# Computational fluid dynamic simulations on liquid film behaviors at flooding in an inclined pipe☆

Jianye Chen [1], Yuan Tang [1], Wei Zhang [2], Yuchen Wang [1], Limin Qiu [1], Xiaobin Zhang [1,*]

[1] Institute of Refrigeration and Cryogenics, Key Laboratory of Refrigeration and Cryogenic of Zhejiang Province, Zhejiang University, Hangzhou 310027, China
[2] China Aerodynamics Research and Development Center, Mianyang 621000, China

ARTICLE INFO

*Article history:*
Received 31 March 2015
Received in revised form 29 May 2015
Accepted 9 July 2015
Available online 18 July 2015

*Keywords:*
Two phase flow
Flooding
Countercurrent flow limitation
Computational fluid dynamic
Liquid film
Inclined pipe

ABSTRACT

The complex liquid film behaviors at flooding in an inclined pipe were investigated with computational fluid dynamic (CFD) approaches. The liquid film behaviors included the dynamic wave characteristics before flooding and the transition of flow pattern when flooding happened. The influences of the surface tension and liquid viscosity were specially analyzed. Comparisons of the calculated velocity at the onset of flooding with the available experimental results showed a good agreement. The calculations verify that the fluctuation frequency and the liquid film thickness are almost unaffected by the superficial gas velocity until the flooding is triggered due to the Kelvin–Helmholtz instability. When flooding triggered at the superficial liquid velocity larger than 0.15 $m\cdot s^{-1}$, the interfacial wave developed to slug flow, while it developed to entrainment flow when it was smaller than 0.08 $m\cdot s^{-1}$. The interfacial waves were more easily torn into tiny droplets with smaller surface tension, eventually evolving into the mist flow. When the liquid viscosity increases, the liquid film has a thicker holdup with more intensive fluctuations, and more likely developed to the slug flow.


## 1. Introduction

In counter-current two phase flow, with the increase of gas flow rates, the flow becomes unstable and finally part or whole of the liquid film reverses flow direction, defined as the onset of flooding. Flooding, as the counter-current flow limitation, is encountered in many industrial devices, such as heat pipe, reflux condenser, packed column and some nuclear reactor accidental scenarios. It often deteriorates the regular operation of devices and has been the subject of numerous investigations in the past decades.

The computational fluid dynamic (CFD) method is a potent tool as it provides more insight into the physics of complex two-phase stratified flows. Volume of fluid (VOF) method with the surface tracking technique and the Eulerian model are widely used to model the flows [1]. Murase *et al.* [2] conducted numerical calculations of the 1/15th scale of pressurized water reactor hot leg with the VOF method. The results underestimated the water flow rates at the upper end of the inclined pipe and overestimated that in the horizontal segment at flooding. Later, they [3] improved the computational grids and schemes to model steam-water flows at 1.5 MPa under pressurized water reactor (PWR) full-scale conditions, and obtained results consistent with the Upper Plenum Test Facility (UPTF) data except in the cases of large steam volumetric flux. Overall, VOF method obtains limited success in modeling flooding because one set of N–S equations are shared by the two phases and the momentum exchange between them ignored.

The Eulerian model solves a set of Navier–Stokes equations for each phase and coupling is achieved through the shared pressure, interphase momentum exchange and energy exchange. The interphase drag force modeling mostly affects the calculation precision when flooding occurs. Wang and Mayinger [4] applied the interfacial friction factor model proposed by Lee and Bankoff [5] to model the UPTF and got a satisfactory result in spite of a little difference of the flow patterns with experimental results in the horizontal leg. Minami *et al.* [6] and Utanohara *et al.* [7] conducted CFD simulations on countercurrent flow in a 1/15th scale of PWR hot-leg model. The required interfacial friction correlations were selected from a combination of the available one-dimensional experimental correlation for annular and slug flows that gave the best agreement with the experimental data.

However, a general geometry-independent model closer to physics and less empirical is a long-term objective for the drag force modeling for flooding simulation. The Algebraic Interfacial Area Density (AIAD) method was adopted by Höhne *et al.* [8–11] to model flooding phenomena in PWR. The results show its success in predicting the transition among different flow patterns. To validate the general usefulness, more work is still required in modeling the flooding in the complex channels where effects of gravity are important.

In this paper, a comprehensive numerical investigation on flooding in inclined pipe is carried out. To verify the accuracy of the numerical method, calculated superficial gas velocities at flooding are compared

☆ Supported by the Major State Basic Research Development Program of China (2011CB706501) and the National Natural Science Foundation of China (51276157).
* Corresponding author.
*E-mail address:* zhangxbin@zju.edu.cn (X. Zhang).

http://dx.doi.org/10.1016/j.cjche.2015.07.012
1004-9541/

with the published experimental data. Emphasis is on the wave characteristics and the flow patterns during flooding. Moreover, impacts of surface tension and liquid viscosity on flooding are investigated.

## 2. Numerical Models

### 2.1. Control equations

In the calculations, we solve the independent mass and momentum conservation equations for the two phases, which have the following form:

$$\frac{\partial \alpha_k \rho_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \boldsymbol{u}_k) = 0 \tag{1}$$

$$\frac{\partial \alpha_k \rho_k \boldsymbol{u}_k}{\partial t} + \nabla \cdot (\alpha_k \rho_k \boldsymbol{u}_k \boldsymbol{u}_k) = -\alpha_k \nabla p_k + \alpha_k \rho_k g + \nabla \alpha_k (\boldsymbol{\tau}_\mathrm{v} + \boldsymbol{\tau}_\mathrm{t}) - \nabla \alpha_k \cdot \boldsymbol{\tau}_{D,k} \tag{2}$$

where $k$ refers to gas (G) or liquid phase (L). The drag force $\boldsymbol{\tau}_\mathrm{D} = |\boldsymbol{\tau}_\mathrm{D,G}| = |\boldsymbol{\tau}_\mathrm{D,L}|$, which is derived from the interfacial shear stress, is most conveniently expressed in terms of the drag coefficient $C_\mathrm{D}$:

$$\boldsymbol{\tau}_\mathrm{D} = \frac{1}{2} C_\mathrm{D} A \rho_\mathrm{LG} |\boldsymbol{u}|^2 \tag{3}$$

where $\rho_\mathrm{LG}$ is the average density, $\boldsymbol{u}$ is the relative velocity and $A$ is the projected area of the control volume in the flow direction. In the AIAD model, $C_\mathrm{D}$ has different correlations in the full range of volume fraction of gas phase, and it allows the detection of the morphological form and the corresponding switching for each correlation from one object pair to another [12]. The asymptotic limits of bubbly and droplet flows are improved by comparing different coefficients. Details can be found in Refs. [8–11].

CFD software Ansys Fluent 14.5 including the two-fluid model was used. Multi-Fluid VOF model for the Eulerian multiphase allows using the sharpening schemes Geo-Reconstruct, compressive, and CICSAM with the Explicit VOF option. This model overcomes some limitations of the VOF model due to the shared velocity and temperature formulation, and is often used for the cases requiring sharp interface treatment between phases. More details can be referred to [13]. The AIAD model, which has successfully predicted the interfacial drag force in the countercurrent flow during flooding [8–11], is adopted in this work.

### 2.2. Turbulence closure

In the counter-current free surface flows, the high velocity gradient at the phase interface will generate high turbulence disturbance in both phases when using differential eddy viscosity models. Hence, turbulence damping is required in the interfacial area to correctly model such flows. For the two-fluid formulation, Egorov *et al.* [12] proposed a symmetric damping procedure based on the standard $\omega$-equation, which provides a solid wall as damping of turbulence in both phases, and is formulated by Wilcox [14] as follows:

$$S_\omega = A_k \Delta n \beta \rho_k \left( B \frac{6\mu_k}{\beta \rho_k \Delta n^2} \right)^2, \quad k = \mathrm{G}, \mathrm{L} \tag{4}$$

which is added as a source to the $\omega$-equation below:

$$\frac{\partial}{\partial t}(\rho\omega) + \frac{\partial}{\partial x_i}(\rho\omega u_i) = \frac{\partial}{\partial x_j}\left( \Gamma_\omega \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + S_\omega. \tag{5}$$

### 2.3. Boundary conditions

Deendarlianto *et al.* [15–17] have carried out lots of flooding experiments in inclined pipes and got abundant experimental data. In this paper, the core computational domain of 1 m length and 16 mm I.D. is modeled as a two-dimensional (2D) structure by reference to the experimental setup [15–17], as shown in Fig. 1. The same 2D model for two-phase pipe flow in a pipe has also been adopted by other researchers [18,19]. Water inlet is conical in order to decrease the disturbance. The inclination is regulated by changing the gravitational direction.

The original grid in the whole domain consists of 95200 cells, as shown in Fig. 2. The grid is doubled to demonstrate the grid independence. The calculated dynamic liquid film thicknesses ($h$) with a void fraction of 50% as the interface are compared with the experimental counterpart when the flow is simulated to the steady state. It indicates that the liquid film wave fluctuates regularly with almost the same amplitudes and frequencies on both computational grids, as depicted in Fig. 3. Therefore, the original grid of 95200 cells is chosen for the following simulations.

The inlets and outlets are velocity inlets and pressure outlets, respectively. The two phases are set as adiabatic and incompressible. At the beginning, an initial liquid film is maintained at a constant superficial liquid velocity ($U_\mathrm{L}$) with a very small gas flow rate $U_\mathrm{G}$. Here, the superficial velocity is

$$U_k = \frac{4V}{\pi D^2} \ (k = \mathrm{G}, \mathrm{L}) \tag{6}$$

where $V$ is the volume flow rate and $D$ is the inner diameter of the tube. Then, $U_\mathrm{G}$ increases with a small gap until partial liquid film reverses its flow, the moment is defined as the occurrence of flooding, and $U_\mathrm{G}$ at this moment is called critical gas velocity ($U_\mathrm{GF}$).

## 3. Result and Discussions

### 3.1. Critical flooding velocities

Fig. 4 illustrates the comparison of calculated $U_\mathrm{GF}$ with the experimental data [17] at different $U_\mathrm{L}$ which as listed in Table 1. Overall, the simulated $U_\mathrm{GF}$ predicted the experimental $U_\mathrm{GF}$ with the accuracy of $\pm 25\%$, which shows the reliability of the simulation method. The deviation between the simulated $U_\mathrm{GF}$ and measured $U_\mathrm{GF}$ is attributed to the different definitions of the onset of flooding. In the experiments [17], the onset of flooding was identified by the maximum airflow rate at which the discharged liquid flow rate is equal to the inlet liquid flow rate. In the numerical simulations, the wave reversal in the close inspection of flow pattern variation was marked as the onset of flooding, the same criterion being also adopted by most of researchers [20–22].

### 3.2. Interfacial characteristics before flooding

Karimi and Kawaji [23] and Vijayan *et al.* [24] performed flooding experiments and reported that the liquid film thickness tended to be little affected by the gas flow before flooding, while it increased sharply when close to flooding. Luo *et al.* [25] also concluded that the effects of gas phase could be neglected at low flow rate through the experiments on inclined plates. In this simulation, a similar trend has been observed. As in Fig. 5, at the upper part of the tube ($x = 0.5$–$0.8$ m), the liquid accelerates downwards after entering the tube and the drag force does little effect on $h$ at $U_\mathrm{G} = 0.08$ and $0.75\ \mathrm{m \cdot s^{-1}}$. When almost approaching flooding ($U_\mathrm{G} = 1.5\ \mathrm{m \cdot s^{-1}}$), the drag force on the liquid film exerted by the gas flow increases significantly, which slows down the liquid film. Therefore, $h$ increases at the upper part ($x = 0.5$–$0.8$ m). It also indicates that all the cases with different gas velocity have a smoother film at $x = 0.5$–$0.8$ m than the lower part, indicating that the interfacial waves progressively slow down when propagating to the liquid outlet, as a result, the amplitude becomes larger. Meanwhile, when close to flooding, as being impeded by large interface