Nuclear Engineering and Design xxx (2016) xxx–xxx

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

CFD prediction of droplet deposition in steam-water annular flow with flow obstacle effects

Haipeng Li *, Henryk Anglart

Royal Institute of Technology (KTH), Stockholm 10691, Sweden

ARTICLE INFO

Article history:
Received 30 May 2016
Received in revised form 17 November 2016
Accepted 23 November 2016
Available online xxxx

Keywords:
Annular two-phase flow
Droplet deposition and entrainment
Flow obstacle
CFD

ABSTRACT

Recent model development on dryout prediction relies on the annular flow modeling, with three fields of gas, droplets and liquid film accounted for. Therefore one unified computational fluid dynamics (CFD) model for annular flow based on Lagrangian particle tracking approach was developed for dryout applications. On the other hand, it is well acknowledged that dryout performance could be improved if flow obstacles are placed in a flow channel. Therefore, to study and predict the dryout, the governing phenomena of droplet deposition and entrainment in annular flow with and without obstacles need to be investigated. The current work tested the CFD model against experimental data from a steam-water flow experiment. Both data with and without obstacles were employed to test the model capability on deposition calculation. The calculated deposition results without obstacles agree reasonably well with both the experimental data and the existing empirical correlations. In case of the flow with obstacles, the calculations also show reasonably good agreement with the experimental data. The current work laid a basis for further work on annular flow model development with dryout capabilities.

1. Introduction

During the simultaneous flow of gas and liquid, several flow patterns can exist, depending on the input flow rates and the physical fluid properties. Among them, annular flow is widely encountered in many engineering applications, such as heat exchangers, boilers, and nuclear reactors. In annular flow, the liquid phase flows partly as a thin liquid film on the channel wall and partly as entrained droplets in the gas core (Hewitt and Hall-Taylor, 1970; Whalley, 1987).

In annular flow with heated walls, the liquid film is depleted by both the entrainment of liquid droplets and by the evaporation. When the liquid film experiences almost complete depletion and then the heated wall is exposed to the gas phase, the heat transfer between the fluid and the channel wall deteriorates, leading to the onset of boiling crisis called dryout, which is a type of critical heat flux (CHF) with high quality (Tong and Tang, 1997).

It is well acknowledged that CHF could be improved if flow obstacles are placed in a flow channel (Pioro et al., 2002). In fuel assemblies of light water nuclear reactors, grid spacers are widely employed to ensure the mechanical integrity, and also to improve the thermal performance. Specifically in a boiling water reactor (BWR), the grid spacers with vanes could enhance the droplet deposition in the fuel rods, to improve the CHF. To study and predict the dryout, the governing phenomena of droplet deposition and entrainment in annular flow with and without obstacles need to be investigated.

Accurate prediction of dryout occurrence is crucial to the design and optimization of BWRs. Compared to the empirical correlations, recent model development on dryout prediction relies on the annular flow modeling, with three fields of gas, droplets and liquid film accounted for (Anglart and Caraghiaur, 2011). Modules with one-dimensional or multi-dimensional capabilities have been included in some system or sub-channel codes, e.g., CATHARE-3 (Emonot et al., 2009; Valette et al., 2009), COBRA-TF (Gluck, 2007), VIPRE-W (Adamsson and Le Corre, 2011), MONA-3 (Hoyer, 1998) and FIDAS (Sugawara and Miyamoto, 1990; Sugawara et al., 1991). Recently Li and Anglart (2015, 2016a, 2016b) have developed the computational fluid dynamics (CFD) capability to model the annular flow for dryout and post-dryout applications. A unified CFD model framework for annular flow was developed, aiming to predict dryout occurrence in nuclear fuel assemblies with spacers. As a first step, it would be important to test the models in an annular flow experiment with obstacles.Concerning droplet deposition, many experiments were conducted previously to obtain the deposition rate and the deposition coefficient, or deposition velocity. For instance, the experiment by Liu and Agarwal

* Corresponding author.
E-mail addresses: haipengl@kth.se (H. Li), henryk@kth.se (H. Anglart).

http://dx.doi.org/10.1016/j.nucengdes.2016.11.021
0029-5493/

Please cite this article in press as: Li, H., Anglart, H. CFD prediction of droplet deposition in steam-water annular flow with flow obstacle effects. Nucl. Eng. Des. (2016), http://dx.doi.org/10.1016/j.nucengdes.2016.11.021

(1974) was widely used for deposition rate correlation development and model validation, where olive oil was used as a liquid dispersed in air flow for aerosol particle study, with particle diameter ranging from 1.4 to 21 μm. In experiments of this type, liquid film flow was deliberately avoided to ease the deposition measurement, which actually is pure mist flow, but not annular flow. The effect of the liquid film on deposition is manifold, including but not limited to, the moving gas core flow boundary and the entrainment and re-deposition of the liquid film. These effects should be accounted for appropriately to obtain the deposition rate in annular flow.

On the other hand, the dryout usually occurs in diabatic flow with evaporation, which requires boiling flow as steam-water flow in BWRs. The experiments for annular flow were usually carried out using air-water flows. The data for steam-water flows on droplet deposition are rather scarce. To the authors' knowledge, the Okawa et al. (2011) experiment provided the latest droplet deposition data in steam-water annular flow with and without an obstacle. In this experiment, a cylindrical tube was concentrically placed in a vertical pipe to investigate the obstacle effects on the droplet deposition.

In the present work, an annular flow CFD model will be employed to simulate the flow in Okawa et al. (2011) experiment, to test the capability of the model for applications with obstacles.

2. Annular flow model

2.1. Liquid film model

As shown in Fig. 1, generally in annular two-phase flow, the liquid phase flows partly as a thin liquid film on the heated wall and partly as droplets in the gas core. The liquid film, especially that in the upstream of the dryout point, is sufficiently thin to safely make the major thin-film assumptions that (1) the flow in the wall normal direction can be reasonably negligible, and (2) the spatial gradients of the dependent variables tangential to the wall surface are negligible compared to those in the wall normal direction.

These assumptions imply that the advection can be treated in the wall tangential direction and diffusion in the wall normal direction, as shown in Fig. 2. As a result, the transport equations for the liquid film can be integrated in the wall normal direction to obtain the two-dimensional equations.

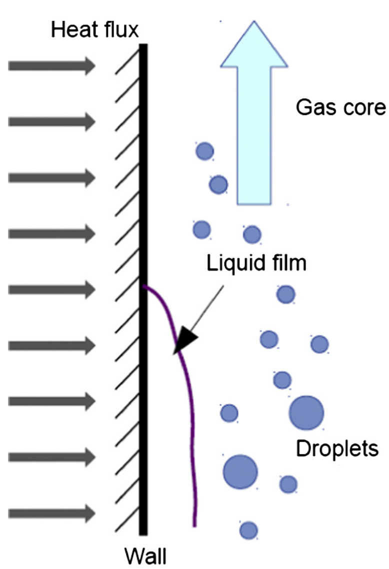

Fig. 1. Schematic of the upward annular flow with depleting liquid film.

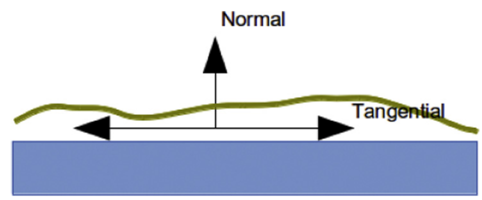

Fig. 2. Schematic of the thin film model and assumed main directions.

All the liquid film properties, which vary across the film thickness, appear as depth-averaged quantities and are in general defined as

$$\bar{\varphi} = \frac{1}{\delta}\int_0^{\delta} \varphi dz \tag{1}$$

where δ is the film thickness, φ is any liquid film property variable, and z is the coordinate for the wall normal direction. For simplicity, the bar is omitted for all the depth-averaged liquid film properties used in the following description.

Then the mass, momentum, and energy equations are integrated in the wall normal direction as

$$\frac{\partial(\rho\delta)}{\partial t} + \nabla_s \cdot (\rho\delta \mathbf{U}_{ff}) = S_\delta \tag{2}$$

$$\frac{\partial(\rho\delta U_{ff})}{\partial t} + \nabla_s \cdot (\rho\delta U_{ff} U_{ff}) = -\delta\nabla_s p + S_{U_{ff}} \tag{3}$$

$$\frac{\partial(\rho\delta h_{ff})}{\partial t} + \nabla_s \cdot (\rho\delta h_{ff} U_{ff}) = S_{h_{ff}} \tag{4}$$

where U_{ff} is the mean film velocity, h_{ff} is the mean film enthalpy, ∇_s is the nabla operator tangential to the surface, ρ is the density, p is the total pressure, and S_δ, $S_{U_{ff}}$ and $S_{h_{ff}}$ are the source terms (Li and Anglart, 2015).

It is noted that the advection terms for all the equations are explicitly described, however, the diffusion and the external sources are modeled as source terms. The liquid film has complex interaction with the gas core flow, which means that corresponding models should be included as source terms to consider all the phenomena of concern.

In diabatic annular flow, the mass sources and sinks for the liquid film are mainly due to the phase change as well as the droplet deposition and entrainment. The source terms for the film momentum have been split into pressure-based part from the tangential gradients in wall normal forces and the stress-based part from the forces in the wall tangential direction. The pressure-based momentum sources include the local gas phase pressure, the hydrostatic pressure, and the pressure due to deposition and entrainment. The stress-based momentum sources consist of gravity force, shear stress, and the forces due to droplet deposition and entrainment. The energy sources include mainly such effects as wall heat transfer, interfacial heat transfer, evaporation, and sources from droplet deposition and entrainment (Li and Anglart, 2015).

2.2. Gas core flow model

In a unified frame for annular flow, the liquid film model should be coupled to simultaneous calculation of the gas core flow including gas and dispersed liquid droplets. The gas core flow can be described using the Eulerian-Eulerian or the Eulerian-Lagrangian methods, both of which have been proven to be capable to capture the governing phenomena. In the current project, the Eulerian-Lagrangian technique is employed, due to the mechanistic