Annals of Nuclear Energy 120 (2018) 155–168

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Numerical investigations on the coupling of the one-group interfacial area transport equation and subcooled boiling models for nuclear safety applications

Abdullah Alali [a,*], Philipp J. Schöffel [b], Joachim Herb [b], Rafael Macian [c]

[a] Nuclear Engineering Department, Jordan University of Science & Technology, P.O. Box 3030, Irbid 22110, Jordan
[b] Gesellschaft für Anlagen- und Reaktorsicherheit (GRS) mbH, 85748 Garching, Germany
[c] Department of Nuclear Engineering, Technische Universität München, 85748 Garching, Germany

ARTICLE INFO

Article history:
Received 2 February 2018
Received in revised form 8 April 2018
Accepted 18 May 2018
Available online xxxx

Keywords:
Subcooled boiling
One-group interfacial area transport equation
OpenFOAM
Wall heat flux partitioning model

ABSTRACT

The one-group interfacial area transport equation (IATE) was coupled to a wall heat flux partitioning model in the framework of two-phase Eulerian approach using the OpenFOAM CFD code for better prediction of sub-cooled boiling flow. The IATE was modified to include the effect of bubble nucleation at the wall and condensation in the bulk region that governs the non-uniform bubble size distribution. To validate the capability of the newly developed OpenFOAM solver, it has been used to simulate the upwards sub-cooled boiling bubbly flow in the DEBORA test facility. Predictions of the gas volume fraction, gas velocity, bubble Sauter mean diameter and liquid temperature profiles were in a good agreement with the experimental data. Simulation results of the DEBORA experiment achieved with the MUSIG model implemented in the ANSYS CFX code in other work have been compared to the simulation results by the IATE model implemented in OpenFOAM to test the competence of the one-group IATE to provide good prediction of subcooled boiling flow parameters. Both approaches were found to provide compatible results.

1. Introduction

The design and safety analysis under normal operation and accident conditions of light water reactors (LWRs) requires a very good understanding of gas-liquid two phase flow and its associated phenomena. In pressurized water reactors (PWRs) the energy generated by the fuel rods is removed by single phase forced convection and, in the hottest fuel assemblies, also by the very efficient sub-cooled boiling heat transfer process.

There are several important safety relevant effects related to the sub-cooled boiling process during normal operation of PWRs. They are induced by the presence of bubbles, which affect the local neutron moderation characteristics and inducing changes in the reactivity of the nuclear reactor (Končar et al., 2005). Another effect is caused by the enhancement, driven by sub-cooled boiling on the surface, of the formation of corrosion products and boron deposition (“crud”) on the cladding surface of the fuel rods, which leads to the so-called Axial Offset Anomaly (AOA), a neutron flux depression at the top of PWR cores. AOA leads to operational difficulties for the reactor (Hawkes, 2004). Furthermore, sub-cooled boiling appears in the downcomer during the reflood phase of a Large Break Loss-of–Coolant Accident (LBLOCA) and reduces the mass flow rate into the core from the emergency core coolant injection to reduce core temperature (Song et al., 2007; Bae et al., 2008).

In the safety analysis of nuclear reactors the flow field is complex and involves two-phase flow which can be represented by field equations and constitutive relations. In the two-fluid model approach adopted in this work, each phase is considered separately in terms of two sets of conservation equations that govern the balance of mass, momentum and energy in each phase (Talley et al., 2011). The constitutive relations representing the phasic interaction terms, the most important characteristics of the two-fluid model formulation, express the transport of mass, momentum and energy through the interface between the phases in term of the interfacial area concentration (IAC) which is related to the geometrical effects of the interfacial structure, and the driving force that characterizes the local transport mechanism of the inter-phase transport (Cheung et al., 2007). Therefore, an accurate estimation of the interfacial area concentration is essential.

* Corresponding author at: Nuclear Engineering Department, Jordan University of Science & Technology, P.O. Box 3030, Irbid 22110, Jordan.
E-mail address: aealali@just.edu.jo (A. Alali).

https://doi.org/10.1016/j.anucene.2018.05.034
0306-4549/

In gas-liquid two-phase vertical flow, the interfacial structure distribution, known as flow regimes, is traditionally classified into five different categories, namely, bubbly, cap, slug, churn-turbulent and annular flow. In most of thermal-hydraulic system analysis codes, the interfacial area concentration is calculated by using an empirical approach based on the two-phase flow regimes and several empirically based regime transition criteria (Mishima and Ishii, 1984; Hibiki and Mishima, 2001). Since these transition criteria are empirical relations, they cannot describe the dynamic nature of the structural changes occurring at the interface and the gradual transition between regimes. Therefore, they are only applicable for steady state and fully developed flow, and only valid for a limited set of flow conditions and geometries (Hibiki and Ishii, 2000).

The internal structure of two-phase flow can be described by the interfacial area concentration that changes with the evolution of the flow due to bubble coalescence and breakup resulting from the interactions among the bubbles and between the bubbles and the turbulent eddies. Therefore, the dynamic change of the interfacial structure could, in principle, be adequately described by a transport equation (Kocamustafaogullari and Ishii, 1995), analogous to the Boltzmann transport equation, that describes the transport of the interfacial area density by an integro-differential equation.

Sub-cooled boiling is characterized by a larger heat transfer capability than single-phase forced convection and contributes to a more efficient cooling of the nuclear fuel rods located in the high power density regions. Unfortunately, the Critical Heat Flux (CHF) limits the heat flux that can be transferred from the rods to the coolant in this manner. The use of computational fluid dynamics (CFD) for nuclear safety application is currently an area of active research, because of the potential it offers to capture local phenomena of safety relevance in regions of the nuclear system, e.g. fuel assemblies, downcomer, Emergency core cooling system (ECCS) injection locations, etc. where they can have an important impact on the safety of the reactors. The detailed descriptions of the flow offered by CFD make them a suitable tool to develop models and carry out simulations in which the coarse models of system codes can be replaced by more mechanistic approaches which require fewer empirically based adjustable parameters. This should provide a higher degree of physical fidelity and modelling accuracy. Heretofore, many researchers worked on the improvement of subcooled boiling modelling using CFD codes. Braz Filho et al. (2016) used FLUENT 14.5 CFD commercial code to model subcooled boiling in conjunction with Kurul and Podowski (1990) boiling model. Yun et al. (2012) examined a mechanistic bubble size model with advanced subcooled boiling model using the STAR-CD 4.12 software. Bae et al. (2010) developed a bubble lift-off mechanistic model in the interfacial area transport equation for the investigation of subcooled boiling using the EAGLE code. To solve the discrepancy observed when modeling subcooled boiling using monodisperse bubble size approach, Krepper et al. (2013) coupled a population balance approach called MUltiple SIze Group (MUSIG) model to a wall boiling model using ANSYS CFX code.

In such a context, the work reported in this paper is dedicated to assess the results of coupling the one-group interfacial area transport equation (IATE) to the wall heat flux partitioning model of Kurul and Podowski (1990) in the framework of two-phase Eulerian approach using the OpenFOAM CFD code for better prediction of sub-cooled boiling flow, which can be also considered as a first step towards the numerical simulation of CHF and the precise prediction of the boiling crisis in computational fluid dynamics (CFD) codes.

2. Model description

2.1. Flow equations

The two-fluid model conservation equations governing the mass, momentum and energy in diabatic flow with heat and mass transfer as used in OpenFOAM are (Ishii and Mishima, 1984)

2.1.1. Continuity equation

$$\frac{\partial}{\partial t}(\alpha_k \rho_k) + \nabla \cdot (\alpha_k \rho_k \vec{U}_k) = \Gamma_{ki} - \Gamma_{ik}, \quad (1)$$

2.1.2. Momentum equation

$$\frac{\partial}{\partial t}(\alpha_k \rho_k \vec{U}_k) + \nabla \cdot (\alpha_k \rho_k \vec{U}_k \vec{U}_k) = -\alpha_k \nabla P_k + \nabla \cdot (\alpha_k (\bar{\bar{\tau}}_k + \bar{\bar{\tau}}_k^{Re})) + \alpha_k \rho_k \vec{g} + \vec{M}_k + \Gamma_{ki} \vec{U}_i - \Gamma_{ik} \vec{U}_k, \quad (2)$$

2.1.3. Energy equation

$$\frac{\partial}{\partial t}(\alpha_k \rho_k H_k) + \nabla \cdot (\alpha_k \rho_k H_k \vec{U}_k) = -\nabla \cdot (\alpha_k (\bar{\bar{q}}_k + q_k^T)) + \alpha_k \frac{Dp_k}{Dt} + \Gamma_{ki} H_i - \Gamma_{ik} H_k + q''_{ki} a_i + \Phi_k, \quad (3)$$

where $\alpha_k, \rho_k, \vec{U}_k$ are respectively, the volume fraction, density and velocity of phase k which can be either liquid (l) or gas (g). $\bar{\bar{\tau}}_k, \bar{\bar{\tau}}_k^{Re}$ are the viscous and Reynolds (turbulent) stresses, respectively. $\vec{M}_k$ is the averaged inter-phase momentum transfer term described in the next sub-section . q_k is the diffusive flux by conduction and the superscript "T" denotes the turbulence enhanced heat flux. q''_{ki} is the interfacial heat flux between the two phases. a_i is the interfacial area concentration. Φ_k is the wall heat source.

The diffusive heat flux by conduction inside a phase k is given by Fourier's law of conduction as

$$q_k = -k_k \nabla T_k \quad (4)$$

where k_k, T_k are thermal conductivity and temperature of phase k, respectively.

2.2. Sub-cooled boiling modelling

Sub-cooled boiling designates the process of evaporation of liquid flowing in contact with a heated solid surface ("wall"), while the bulk liquid temperature is lower than local saturation temperature. When the wall temperature exceeds the local liquid saturation temperature, micro-cavities distributed over it, called nucleation sites, activate the formation of vapor bubbles by becoming centers around which steam can accumulate forming bubbles. They grow until they reach a critical size. At this point, the bubbles slide along the heated surface while continuing to grow until they become large enough that buoyancy forces overcome surface tension forces and the bubbles can leave the wall and migrate laterally towards the sub-cooled bulk liquid. There they condensate, releasing their energy to the bulk liquid. Following this description, sub-cooled boiling is modelled as a combination of phase change due to bubble generation near the heated wall described by evaporation rate Γ_{gl}, and phase change due to condensation of the generated bubbles after departure from the wall induced by the subcooled liquid in the bulk and is described by the condensation rate Γ_{lg}.

The mass transfer rate per unit volume due to condensation in the bulk subcooled liquid Γ_{lg} is given by