ARTICLE IN PRESS

Progress in Nuclear Energy xxx (xxxx) xxx–xxx

Contents lists available at ScienceDirect

Progress in Nuclear Energy

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

Simplified two-group two-fluid model for three-dimensional two-phase flow Computational Fluid Dynamics for vertical upward flow

Takashi Hibiki[a], Joshua P. Schlegel[b], Tetsuhiro Ozaki[c,d], Shuichiro Miwa[d], Somboon Rassame[e,*]

[a] School of Nuclear Engineering, Purdue University, 400 Central Drive, West Lafayette, IN 47907-2017, USA
[b] Department of Mining and Nuclear Engineering, Missouri University of Science and Technology, 301 W 14th St, Rolla, MO 65409, USA
[c] Tokai Works, Nuclear Fuel Industries, Ltd., 3135-41, Muramatsu, Tokai, Naka, Ibaraki 319-1196, Japan
[d] Division of Energy and Environmental Systems, Graduate School of Engineering, Hokkaido University, North 13, West 8, Kita, Sapporo, Hokkaido 060-8628, Japan
[e] Department of Nuclear Engineering, Faculty of Engineering, Chulalongkorn University, Wangmai, Patumwan, Bangkok 10330, Thailand

ARTICLE INFO

Keywords:
Interfacial force
Interfacial area transport equation
Two-fluid model
CFD
V & V

ABSTRACT

Recent progress in nuclear thermal-hydraulics simulations has been largely focused on coupling with other computational packages, improved closure models for subcooled boiling and for bubbly flows, and the development of higher-fidelity simulation capabilities (Kulesza et al., 2016). While high-fidelity 3D simulation is important for model validation, scientific understanding, and some design calculations, it can be prohibitively expensive for system design applications or applications involving large geometries. Thus, there is also a need for practical, simplified approaches for those applications. The two-fluid model strikes a balance between detail and computational resources, but requires the accurate specification of several key constitutive models. These include (1) interfacial forces, (2) interfacial area concentration, (3) two-phase turbulence, and (4) wall and bulk boiling and condensation. In many modern CFD packages, uncertainties in the local interfacial area concentration can have strong effects on the ability to predict the other key parameters. This paper demonstrates that the drag force in 3D CFD can be formulated in much the same way as in 1D system analysis codes and that this approach can be used to formulate a model for interfacial area concentration. The method is also applied to two-group approaches to consider the difference in transport properties for different bubble size classes. This approach may open a method to calculate the interfacial forces without the need for interfacial area transport equations. This reduces the number of differential equations and avoids the modeling challenges associated with bubble breakup and coalescence kernels and the need to specify the inlet interfacial area concentration *a priori*. The new method is expected to decouple the effects of interfacial area uncertainty and calibrated coefficients, and should provide reasonable local bubble diameters for both group-1 and group-2 bubbles. The approaches proposed in this study are applicable to two-phase flow simulations in rather simple geometries such as upward two-phase flow in vertical channels. In view of many applications for upward two-phase flow in vertical channels, including nuclear reactor systems, the proposed methods are considered useful.

1. Introduction

In 2010 the Consortium for Advanced Simulation of Light Water Reactors (CASL) was established to provide advanced modeling and simulation capability for commercial nuclear reactors (Kulesza et al., 2016). The consortium is developing "*coupled, higher-fidelity, usable modeling and simulation capabilities*" to improve the prediction of light water reactor operation and safety performance-defining phenomena (Kulesza et al., 2016). The consortium has identified 10 major "*Challenge Problems*" which impact power uprate, higher burnup, life extension and safety. They are categorized into two major groups. The first group is "*Operational Challenge Problem*" such as power shift, CRUD-induced localization corrosion, grid-to-rod fretting failure, pellet-clad interaction, and fuel assembly distortion. The second group is "*Safety Challenge Problems*" including departure from nucleate boiling, cladding integrity during loss of coolant accidents, cladding integrity during reactivity insertion accidents, reactor vessel integrity, and reactor internals integrity (Kulesza et al., 2016). CASL has identified six focus areas to accomplish their goal, namely (1) advanced modeling applications, (2) physics integration, (3) radiation transport methods, (4) materials performance and optimization, (5) validation and uncertainty quantification, and (6) thermal-hydraulics methods (Kulesza et al., 2016). Among these, the role of thermal-hydraulic methods is significant. Turinsky and Kothe (2016) pointed out "*CFD R&D has focused*

---

[*] Corresponding author.
E-mail address: somboon.ra@chula.ac.th (S. Rassame).

https://doi.org/10.1016/j.pnucene.2017.12.003
Received 26 September 2017; Received in revised form 13 December 2017; Accepted 18 December 2017
0149-1970/

Please cite this article as: Hibiki, T., Progress in Nuclear Energy (2017), https://doi.org/10.1016/j.pnucene.2017.12.003

on improvement in closure models for subcooled boiling and bubbly flow, and the formulation of robust numerical solution algorithms".

High fidelity 3D (Three-Dimensional) CFD (Computational Fluid Dynamics) simulation is important to simulate various thermal-hydraulic two-phase flow phenomena in nuclear related components, however there are limitations in high-fidelity approaches to two-phase flows. High fidelity numerical approaches for multiphase flows generally involve some form of interface tracking method. These methods are currently limited by computer technology to simulations involving about 100 bubbles. It is estimated that such approaches will be able to predict flows with up to 1% void fraction in 50 years, given the current rate of advancement in computer technology. By comparison the void fraction in a typical PWR is up to 1% during normal operating conditions (Westinghouse, 2011) and the void fraction in Boiling Water Reactors (BWRs) or steam generators can be 70% or higher.

Given the impracticality and high cost of such prediction methods for most industrial applications, a more reasonable approach is necessary for a wide range of engineering design calculations. Among the available two-phase flow analysis techniques, the two-fluid model has been well-developed and is one of most practical two-phase flow models (Ishii and Hibiki, 2010). The two-fluid model is composed of a set of mass, momentum and energy transport equations for each phase. Due to the complex nature of two-phase flow, two-phase flow CFD has not been well-validated. To develop practical high-fidelity two-phase flow CFD using the Reynolds averaging and the two-fluid model, several key constitutive models should be accurately provided. They are (1) interfacial forces, (2) interfacial area concentration, (3) two-phase flow turbulence, and (4) wall and bulk boiling and condensation. These areas are illustrated in Fig. 1 (Chuang and Hibiki, 2015).

The interfacial forces are key to simulating three-dimensional phase distributions accurately. Chuang and Hibiki (2017) provided comprehensive review on the available interfacial forces for two-phase flow CFD. The interfacial area concentration provides an important length scale to characterize two-phase flow and is closely related to the interfacial transfer terms. Lin and Hibiki (2014) provided an extensive review on the available interfacial area data to benchmark interfacial area constitutive models, and Chuang and Hibiki (2015) provided a state-of-the-art review of two-phase flow CFD using the interfacial area transport equation. Vaidheeswaran and Hibiki (2017) provided an extensive review on the available bubble-induced turbulence models for vertical bubbly flow, but the progress of the two-phase flow turbulence modeling is limited due to insufficient experimental data. Hibiki and Ishii (2003) and Brooks and Hibiki (2015) developed constitutive models for active nucleation site density, bubble departure frequency and bubble departure diameter, which enhance the prediction accuracy of wall nucleation source terms.

As discussed above, accurate modeling of the interfacial forces is indispensable for successful three-dimensional two-phase flow simulation. One of the most important interfacial forces is the drag force. The interfacial drag force determines the relative velocity between the phases and plays a large role in determining the phase volume fractions (Ishii and Zuber, 1979). It is commonly formulated in terms of local interfacial area concentration. Therefore, prediction uncertainty for local interfacial area concentration directly affects the accuracy of the drag force and phase volume fractions. In current three-dimensional two-phase flow simulation, the interfacial area transport equation calculates the dynamic change of local interfacial area concentration, including the effect of bubble coalescence and breakup. The prediction of local interfacial area concentration in the simulation may be reasonable if the correct inlet interfacial area concentration is given (Lee et al., 2013; Sharma et al., 2017). In other words, the prediction accuracy of local interfacial area concentration is heavily dependent on the inlet boundary value provided by the user, not only the accuracy of the constitutive models. To implement a robust drag force formulation in one-dimensional thermal-hydraulics system analysis codes, Andersen and Chu (1981) formulated the one-dimensional drag force using drift-flux correlations to eliminate the dependence of the one-dimensional drag force on the interfacial area concentration.

This study demonstrates that the Andersen and Chu (1981) approach developed for one-dimensional two-phase flow simulation can be applied to three-dimensional two-phase flow simulation. This study also shows local interfacial area concentration consistent with the local drag formulation. The local interfacial area concentration may be used in calculating other non-drag forces. This approach may open up a way to calculate the interfacial forces without local interfacial area transport equation, which has not been well-modeled.

2. Two-group two-fluid model and generalized interfacial drag force formulation

Bubble transport characteristics are dependent on bubble size. For example, lift force pushes small bubbles to the wall of the channel and large bubbles towards the channel center, respectively, resulting in wall and core peaking in void fraction distribution (Hibiki and Ishii, 2007). In order to perform spatially resolved two-phase flow simulation, a conventional two-fluid model with a single bubble size obtained by averaging all bubble sizes may not be sufficient. Multiple Size Group (MuSiG) models (Lo, 1996; Frank et al., 2005; Liao et al., 2011) and

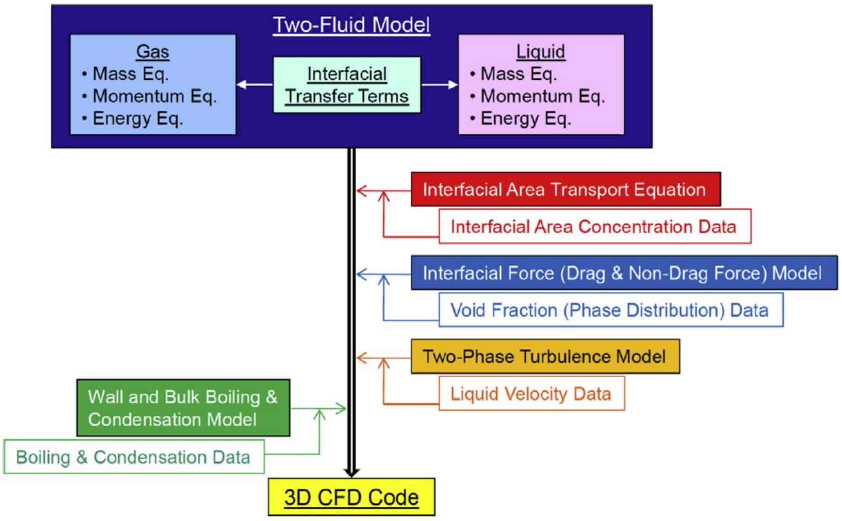

Fig. 1. Key constitutive models and local data for 3D two-phase CFD (Chuang and Hibiki, 2015).