Annals of Nuclear Energy 111 (2018) 311–316

Contents lists available at ScienceDirect

Annals of Nuclear Energy

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

Modelling horizontal two-phase flows using generalized models

Thomas Höhne *, Paul Porombka

Helmholtz-Zentrum Dresden-Rossendorf (HZDR) – Institute of Fluid Dynamics, Bautzner Landstr. 400, D-01328 Dresden, Germany

ARTICLE INFO

Article history:
Received 16 December 2016
Received in revised form 28 August 2017
Accepted 13 September 2017

Keywords:
CFD
AIAD
Two-phase flow
Subgrid wave turbulence

ABSTRACT

Gas–liquid two-phase flows in the hot leg of a pressurized water reactor (PWR) under hypothetical accident scenarios have received special attention in nuclear reactor safety research. The numerical simulation of such flows can be performed using phase-averaged multi-fluid models, such as the two-fluid approach, or non-phase-averaged alternatives. The method shown in this paper within the two-fluid framework is the Algebraic Interfacial Area Density (AIAD) model. It features the macroscopic blending between different interfacial area density and momentum transfer models depending on the local flow morphology. The work presented here strives to improve the turbulence modelling for free surface flows by introducing a model for sub-grid size waves induced by Kelvin-Helmholtz instability. A first CFD validation of the methodology is done by means of horizontal, adiabatic, stratified flow data from the WENKA facility. More verification and validation of the approach is required – more CFD grade experimental data is the key to further validation.

1. Introduction

During the last decades, the application of three dimensional (3D) computational fluid dynamics (CFD) codes in the field of nuclear reactor safety has clearly gained in importance. Accordingly, the use of 3D-CFD codes to predict horizontal two phase flow phenomena in nuclear applications, such as slugging, counter-current flow limitation and pressurized thermal shock (PTS), tends to be of growing interest. The availability of detailed 3D information on the respective phenomenon is establishing as a new standard in the reactor safety analysis. Empirical and geometry-dependent closure laws which are a prerequisite in 1D system codes, can be replaced by physically more fundamental closure laws in 3D-CFD. In this sense, CFD can be geometry-independent and thus more flexible than 1D system codes.

In Eulerian simulation approaches for two-phase flows, the information on the interfacial structure, such as surface ripples, waves or turbulent fluctuations, is lost by phase averaging the governing equations. Consequently, the effect of the non-resolved scales has to be accounted for by a model. More specifically, a model for the interfacial area density is required to correctly predict the interfacial transfers. The larger waves can be simulated, but the detailed structure of interacting boundary layers of the separated continuous phases cannot be resolved. Instead, its influence on the average flow must be included by a physical model.

The mass, momentum and heat transfer across the phase boundary depends on its small-scale structures which are not resolved in an Eulerian approach. Thus, in the two-fluid framework the interfacial momentum transfer is modelled using correlations for the interfacial drag. Due to a lack of appropriate models, drag correlations for dispersed flow or correlations inherited from 1D codes were applied to the interfacial momentum transfer in the past. Such approaches use simplifying assumptions which do not reflect the underlying phenomena at the interface.

Direct numerical simulations (Fulgosi et al., 2003) confirmed the widespread assumption that the stratified phase boundary acts like a moving solid wall on the gas-side velocity and turbulence fields. This wall may have a roughness in case of waves. These findings are incorporated in the interfacial momentum transfer model of the Algebraic Interfacial Area Density (AIAD) framework (Höhne and Vallée, 2010). The AIAD framework enables blending of the models for arbitrary quantities such as interfacial area density and interfacial drag based on the local flow morphology.

The phasic averaging in the Eulerian approach also affects the turbulent motion at the phase boundaries. Waves which are generated by the Kelvin-Helmholtz instability typically have a length scale below the grid size, yet they contribute to the turbulence level in the interface region. The small-wave contribution to the turbulence kinetic energy therefore has to be account for by a model. Here, a previously (Höhne and Hänsch, 2015) proposed model for this turbulence generating mechanism is validated against experimental data from the WENKA test facility (Stäbler, 2006).

---

* Corresponding author.
E-mail address: t.hoehne@hzdr.de (T. Höhne).

http://dx.doi.org/10.1016/j.anucene.2017.09.018
0306-4549/© 2017 Elsevier Ltd. All rights reserved.

This paper is organized as follows. Firstly, the AIAD framework is illustrated in Section 2, which is followed by descriptions of the model enhancements for the interfacial drag and turbulence in Sections 2.1 and 2.2, respectively. The simulation results are discussed in Section 3. Concluding remarks are given in the last section.

2. Algebraic Interfacial Area Density Model (AIAD)

The AIAD model enables the recognition of the flow morphology and the consistent blending of arbitrary closure relations. It has been defined previously by Höhne and Vallée (2010), so that only a brief description is given here.

The central idea of the AIAD model is to define a set of three algebraic weighting functions that depend on the local volume fraction. The three flow morphologies under consideration are: dispersed gas in continuous liquid phase (B), dispersed liquid in continuous gas phase (D) and the continuous gas-liquid free surface region (FS). Each flow morphology is represented by a corresponding blending function:

$$f_D = \left[1 + e^{a_D(\alpha_L - \alpha_{D,\text{limit}})}\right]^{-1} \tag{1}$$

$$f_B = \left[1 + e^{a_B(\alpha_G - \alpha_{B,\text{limit}})}\right]^{-1} \tag{2}$$

$$f_{FS} = 1 - f_D - f_B \tag{3}$$

here, $a_D,\ a_B = 70$ denote blending coefficients for the droplet and bubble regimes, respectively and $\alpha_{D,\text{limit}},\ \alpha_{B,\text{limit}} = 0.3$ are the critical volume fractions for the corresponding regimes. The model has a low sensitivity to changes in the blending coefficients $a_D,\ a_B$. Optimum values for numerical stability have been determined in earlier studies on free surface flow (Höhne and Mehlhoop, 2014) and are adopted here without further tuning or adaption to the geometry.

2.1. Modelling the free surface drag

In the Eulerian two-fluid model a closure relation for the momentum exchange at the phase boundaries is required. In this study the sole mechanism for momentum exchange considered is the drag force. The standard quadratic drag law for the drag force per unit volume is assumed

$$|F_D| = C_D a \rho |\boldsymbol{U}|^2 \tag{4}$$

where C_D is the total drag coefficient, a is the interfacial area density, $\boldsymbol{U}$ denotes the gas-liquid relative velocity and the mixture density

$$\rho = \alpha_G \rho_G + \alpha_L \rho_L \tag{5}$$

is calculated from the gas density ρ_G, the liquid density ρ_L and the corresponding volume fractions α_G, α_L, respectively.

The drag law (4) has to be provided with a model for the unknown drag coefficient C_D. By means of the AIAD blending functions, different C_D can be used depending on the flow morphologie. In bubbly and droplet regimes, a constant drag coefficient $C_{D,B} = C_{D,D} = 0.44$ is applied, assuming spherical fluid particles and Newton drag. If required, Reynolds number dependent drag correlations can be used in these regimes within the AIAD model as well. In the free surface regime the spherical particle assumption with a constant drag coefficient is dropped in favour of a more realistic drag coefficient model which has been introduced and qualitatively validated in a previous work by Höhne and Hänsch (2015). It is built around the assumption that momentum exchange at a large gas-liquid interface with unit normal vector

$$\mathbf{n} = -\frac{1}{|\nabla\alpha|}(\partial_x\alpha, \partial_y\alpha, \partial_z\alpha)^{\mathrm{T}} \tag{6}$$

is similar to the shear stress at a solid wall boundary. Consequently, the drag force in the free surface regime is expressed as a shear force

$$F_W = \tau_i A = F_D. \tag{7}$$

Expressing the interfacial shear stress τ_i as outlined by Höhne and Hänsch (2015) leads to an expression for the free surface drag coefficient

$$C_{D,FS} = \frac{(\alpha_L t_{w,L} + \alpha_G t_{w,G})}{\rho |\boldsymbol{U}|^2}. \tag{8}$$

2.2. Sub-grid wave turbulence

Phase-averaged CFD methods, such as the two-fluid model, usually neglect waves below the grid size. Such waves may be formed for instance by Kelvin-Helmholtz instabilities. Despite that, the effect of these sub-grid size waves on liquid side turbulence kinetic energy (TKE) can be noticeably large. (Brocchini and Peregrine, 2001) performed a detailed analyses of strong turbulence at air-water free surfaces which is followed here. The behavior of a gas-liquid interface depends both on gravity and surface tension forces, it can therefore be characterized by the corresponding dimensionless groups, i.e. the turbulent Froude number $Fr = q/(2gL)^{1/2}$ and the Weber number $We = q^2 L\rho/2\sigma$. Here, $q = \sqrt{2k}$ represents a turbulent velocity scale, L is a turbulent length scale and σ denotes the interfacial tension coefficient. The effect of sub-grid size waves is considered in terms of a critical Fr,We parameter space where the surface is no longer smooth and does not break up completely. A corresponding volumetric source term for the sub-grid wave turbulence was formulated and added to the liquid side TKE transport equation (Höhne and Hänsch, 2015).

3. Simulation results and comparison with WENKA experiment

Experimental data of countercurrent air-water flow in a horizontal rectangular channel of the WENKA facility (Stäbler, 2007) is used for validation. The experimental setup represents a simplified model of a hot leg injection in pressurized water reactors. It consists of closed water- and open air loop feeding the rectangular, horizontal test section as depicted in Fig. 1.

Water enters the test section from the left while air enters from the right side. The liquid level at the inlet y_0 is realized with an adjustable splitter plate. At the indicated measurement positions MP1 and MP2, the profiles along the y axis of the following variables have been measured in both phases: velocities $\overline{u}, \overline{v}$, the root mean square (rms) velocity fluctuations u_{rms}, v_{rms}, volume fraction α and rms Reynolds stress component $\tau_{T,x,y}$. Additionally the measured mean wave amplitude y_δ is available at MP1 and MP2. More details on the experimental facility and a detailed description of the measurement methods are given in Stäbler (2007, 2006).

The selected measurement points 3 and 23 lie within the stratified-wavy flow regime and correspond to positions MP1 and MP2. The flow parameters in the experiment are given in the table below (Table 1).

Here, the bulk inlet velocities of the liquid and gas phase are indicated as U_{in}^{L} and U_{in}^{G}, respectively, y_0 denotes the inlet water level and Re_d^L, Re_d^G are the liquid and gas phase Reynolds numbers based on the hydraulic diameter. The Froude number at the water inlet $Fr_0 = U_{\text{in}}^{\text{L}}/\sqrt{gy_0}$ indicates supercritical flow conditions with rising liquid level in downstream direction.