Nuclear Engineering and Design 305 (2016) 371–377

Contents lists available at ScienceDirect

Nuclear Engineering and Design

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

Grid studies for the simulation of resolved structures in an Eulerian two-fluid framework

Friederike Gauss *, Dirk Lucas, Eckhard Krepper

Institute of Fluid Dynamics, Helmholtz-Zentrum Dresden – Rossendorf, Dresden, Germany

HIGHLIGHTS

- Elaborated Eulerian two-fluid methods may predict multiphase flow with large differences in interfacial length scales.
- A study on the grid requirements of resolved structures in such two-fluid methods is presented.
- The two-fluid results are only little dependent on the grid size.
- The results justify the resolved treatment of flow structures covering only few grid cells.
- A grid-dependent limit between resolved an modeled structures may be established.

ARTICLE INFO

Article history:
Received 25 January 2016
Received in revised form 29 April 2016
Accepted 4 June 2016

JEL classification:
W. Miscellaneous

ABSTRACT

The influence of the grid size on the rise velocity of a single bubble simulated with an Eulerian two-fluid method is investigated. This study is part of the development of an elaborated Eulerian two-fluid framework, which is able to predict complex flow phenomena as arising in nuclear reactor safety research issues. Such flow phenomena cover a wide range of interfacial length scales. An important aspect of the simulation method is the distinction into small flow structures, which are modeled, and large structures, which are resolved. To investigate the requirements on the numerical grid for the simulation of such resolved structures the velocity of rising gas bubbles is a good example since theoretical values are available. It is well known that the rise velocity of resolved bubbles is clearly underestimated in a one-fluid approach if they span over only few numerical cells. In the present paper it is shown that in the case of the two-fluid model the bubble rise velocity depends only slightly on the grid size. This is explained with the use of models for the gas–liquid interfacial forces. Good approximations of the rise velocity and the bubble shape are obtained with only few grid points per bubble diameter. This result justifies the resolved treatment of flow structures, which cover only few grid cells. Thus, a limit for the distinction into resolved and modeled structures in the two-fluid context may be established.

1. Introduction

One important aspect of nuclear reactor safety (NRS) research is the development of reliable simulation tools, which include the prediction of multi-phase flow phenomena such as steam injection into pools or steam bubble entrainment into sub-cooled liquids by impinging jets, e.g. in case of emergency core cooling injection into a partially uncovered cold leg (Lucas et al., 2011).

These multiphase flow phenomena cover different flow patterns, ranging from small structures, like small air bubbles in water, to large structures, like free surfaces. In many realistic scenarios, small and large interfacial structures exist simultaneously. In the simulation context, the distinction into large and small structures depends on the grid size. Large structures cover several grid cells, whereas small structures are in the range of grid size. Thus, the simulation of flows with both large and small interfaces requires extended models, which distinguish between the scales and take into account the different flow patterns.

Typically, for the simulation of flow structures with large interfaces, one-fluid simulation models like Volume of Fluid (VoF) methods, introduced by Hirt and Nichols (1981) or Level Set (LS) methods (Sussmann et al., 1994) are used. One-fluid methods, also classified as one-field or single-field methods, calculate one velocity field together with a transport equation for a coloring function which characterizes the interface. These methods need a high

* Corresponding author at: Helmholtz-Zentrum Dresden – Rossendorf, Institute of Fluid Dynamics, Bautzner Landstrasse 400, 01328 Dresden, Germany. Tel.: +49 351 260 31 79.

E-mail address: f.gauss@hzdr.de (F. Gauss).

http://dx.doi.org/10.1016/j.nucengdes.2016.06.009
0029-5493/© 2016 Elsevier B.V. All rights reserved.

number of grid points to resolve the interfacial structures. This limits the applicability of one-fluid methods to the simulation of large-scaled structures, as a resolution of small interfaces is not suitable.

Otherwise, Eulerian two-fluid methods compute for each phase an own velocity field. The considered variables are ensemble averaged values. Both phases are present simultaneously in every grid cell with certain probabilities given by the phasic volume fractions. Due to the averaging of the variables small-scaled interfaces are not simulated in the two-fluid method. Interactions between the phases, which depend on these interfaces, like momentum transfer, have to be considered by closure models. As the small structures are not resolved, two-fluid models are suitable for the simulation of dispersed flows.

In the two-fluid context, large interfacial structures are not sharply defined as in the one-fluid approach, but smeared due to the averaging of the variables. Here, large interfaces may be characterized by sharp gradients of the volume fractions (Hänsch et al., 2012). In order to counteract a numerical smearing of the gradients during the solution of the volume fraction transport equation, certain techniques have to be applied to keep the gradients and hence the interfaces sharp. Zwart et al. (2003) propose a compressive interpolation technique in the volume fraction continuity equation, whereas Hänsch et al. (2012) introduce a clustering force; Strubelj et al., (2009) solve an additional equation, which acts as an artificial compression, right after the solution of the continuity equation.

Thus, extended Eulerian two-fluid methods with interface sharpening methods are capable of simulating complex multiphase flow phenomena with co-existing large-scaled and dispersed small-scaled structures. One recently developed approach is the GENTOP (GENeralized TwO-Phase) concept (Hänsch et al., 2012). It takes into account at minimum three phases: a poly-dispersed gas phase for the small-scaled bubbles, a continuous gas phase for the large-scaled structures, and a continuous liquid phase. The poly-dispersed gas phase is modeled based on the inhomogeneous Multiple Size Group (iMUSIG) approach (Krepper et al., 2008). Large-scaled interfaces between the continuous gas and continuous liquid phase are detected by a surface function, which is based on the comparison of the gradient of the continuous gas volume fraction with a certain grid size dependent critical value. For the interface sharpening, a clustering force is introduced as an additional source term in the momentum equation. Transitions between dispersed bubbles and large gas structures can be modeled by coalescence and breakup processes basing on the included population balance.

In the GENTOP concept, the distinction between small (modeled) and large (resolved) gas structures is based on the grid size, where structures with a length scale (e.g. bubble diameter) larger than 4 typical mesh cell sizes Δx are statistically resolved, i.e. resolved in the two-fluid context, where the volume fractions give only probabilities for the presence of the according phase. It is mentioned in Hänsch et al. (2012), that this value of $4\Delta x$ 'has to be checked carefully'.

One crucial point in the GENTOP concept is the prediction of the correct flow behavior of the resolved structures, especially for the structures near the threshold value mentioned above. The correct flow behavior may be the correct rise velocity and bubble shape in the simulation of a large resolved rising bubble, for example. This leads to the question of a sufficient grid resolution for the large structures.

It is known from literature, that for one-fluid methods, a certain number of grid points is needed to resolve the interfacial details in order to obtain reasonable results as the interfacial forces are directly calculated. Otherwise, in the two-fluid simulations, the interfacial forces are reflected by models. These models, e.g. for the drag force, determine the relative velocity between the two phases. This leads to the assumption, that the two-fluid simulation results are only little dependent on a numerical parameter like the grid size and that only few grid points are needed for the bubble resolution.

The present paper investigates this question of sufficient grid resolution for the case of a single rising bubble in stagnant fluid. This flow phenomenon is well studied in order to understand the characteristics of bubbles. Some fundamental work can be found e.g. in Clift et al. (1978), Fan and Tsuchiya (1990) and Bhaga and Weber (1981).

Most of the literature investigating the grid resolution for the simulation of a single rising bubble deals with one-fluid methods. Andersson et al. (2012) point out, that for VoF methods 'typically, about 20 cells/diameter will be needed in order to obtain satisfactory resolution of a spherical bubble or drop'. In Svihla and Xu (2006), a 2D bubble with 2 mm diameter was resolved with about 20 cells/diameter. It is mentioned, that 'the calculation is somewhat imprecise since the grid resolution for the bubble is fairly coarse'. In Badreddine et al. (2015) a grid study is performed for the rise of 1 mm and 2 mm air bubbles in stagnant water, showing that a number of at least 20 grid cells per bubble diameter is needed to obtain agreement with experimental data concerning bubble shape and terminal rise velocity. In Engberg et al. (2014), a much higher number of 80 cells is mentioned to resolve a 3D toluene bubble with 5 mm diameter in a Level Set simulation, and even about 300 cells, if Marangoni convection is taken into account. Amaya-Bower and Lee (2010) show, that a bubble resolution with 40 grid cells per diameter gives reasonable accuracy in the 2D simulation of bubbles for with a Lattice Boltzmann Method based on the Cahn–Hilliard diffusive interface approach.

On the other hand, there are only few statements concerning the grid requirements for the bubble simulation with an Eulerian two-fluid method. Strubelj et al. (2009) show results for a two-fluid simulation of a 2D bubble at low Reynolds numbers using non-realistic fluid properties. The bubble is resolved with high numbers of 40, 80 and 160 cells/diameter. Here, the comparison of the bubble rise velocity and the bubble circularity shows practically no grid dependency, which indicates, that a smaller number of grid points could be sufficient.

This paper provides grid studies for calculations with the Eulerian two-fluid method performed on successively refined grids. Three different bubble sizes are considered. The results are evaluated based on the terminal bubble rise velocity and the bubble shape. The aim is to show that the two-fluid method is only slightly grid dependent and that good approximations may be obtained with only few grid points for the bubble resolution. Based on the results, a grid-size dependent limit between the small modeled and large resolved structures in the GENTOP concept may be established.

2. Physical and numerical basics

The governing equations for the description of the flow of Newtonian fluids are the Navier–Stokes equations, describing the conservation of mass and momentum.

In the Eulerian two-fluid approach each of the phases has its own velocity field. Thus, two sets of equations are solved for the phases $j = 1, 2$:

$$\frac{\partial}{\partial t}\left(\alpha_j \rho_j\right) + \nabla\left(\alpha_j \rho_j \vec{U}_j\right) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}\left(\alpha_j \rho_j \vec{U}_j\right) + \nabla \cdot \left(\alpha_j \rho_j \vec{U}_j \times \vec{U}_j\right) = \nabla \cdot \left[\alpha_j \mu_j \left(\nabla \vec{U}_j + \left(\nabla \vec{U}_j\right)^T\right)\right] - \alpha_j \nabla p + \alpha_j \rho_j \vec{g} + \vec{M}_j + \vec{S}_j \tag{2}$$