Computers & Fluids 113 (2015) 106–111

Contents lists available at ScienceDirect

Computers & Fluids

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

# Multifield hybrid approach for two-phase flow modeling – Part 1: Adiabatic flows

Romain Denèfle [a,*], Stéphane Mimouni [a], Jean-Paul Caltagirone [b], Stéphane Vincent [b]

[a] EDF R&D, 6 quai Watier, 78401 Chatou, France
[b] I2M, Université Bordeaux, UMR 5295, 33607 Pessac, France

ARTICLE INFO

*Article history:*
Received 3 December 2013
Accepted 17 July 2014
Available online 31 July 2014

*Keywords:*
Multifield model
Surface tension
Interface sharpening
Bubble rising
Plume
Model coupling

ABSTRACT

As a step towards a hybrid approach for the simulation of two-phase flow transitions, the objective of the present work is to check out the capability of a numerically effective hybrid model. The multifield hybrid approach for two-phase flow modeling consists in dealing separately with the small and spherical bubbles, treated with a dispersed approach, and with the large and distorted ones, which interface is located. These two approaches are coupled through mass transfer terms. The overall method relies on two existing blocks, consisting in a set of averaged models dedicated to dispersed bubbles and another one dedicated to interface locating, which have already been validated and have given a reasonable agreement with experimental data.

The main outcome is the simulation of a three field case with a complete set of coupling terms between the two gas fields. This approach is used to simulate the experiment of Castillejos, a bubble plume in a water tank presenting a wide range of bubble diameter.



## 1. Introduction

There are several approaches for two-phase flow modeling, describing the interfaces either with a dispersed or a located point of view. Bubbly flows are often modeled with an Eulerian dispersed description, within the two-fluid model of Ishii [1]. The averaged momentum balance equation is in this case closed with a set of interfacial forces such as drag, lift, virtual mass and turbulent dispersion. These forces rely on empirical or statistical correlations making assumptions on the bubbles mean diameter and shape. On the other hand, large interface flows such as slugs or free surfaces are mostly simulated through located approaches such as front tracking [2], Level-set [3,4] or Volume of Fluid [5] with an Eulerian point of view or Lagrangian grid methods [6] with a Lagrangian point of view. All these methods aim at calculating the local characteristics of the interface, such as the curvature and the normal vector, to model the interfacial transfer in the momentum equation.

New approaches are explored to simulate more accurately the transition regime between bubbly and separated flows. The concept of a four field and two-fluid model has been presented and studied over the last decade [7,8]. Each phase is decomposed into a continuous and a dispersed field, resulting in a four field system of mass, momentum and energy equations. This kind of approach requires the set of mass transfer terms between the continuous and the dispersed fields of the same physicochemical phase. A spatial cutting length is dividing a phase between unresolved structures that are modeled and the larger ones that are simulated. This concept allows the simulation of a wide range of two-phase flow regimes with both a good accuracy on the behavior of the most distorted interfacial structures, and less CPU consuming than the direct simulation of every two-phase scales. If the dispersed fields are commonly dealt with an Eulerian point of view, several methods can be used to locate the interface between the liquid and gas continuous fields.

We present a hybrid multifield approach based on this four field concept. It consists in modeling the two-phase flow with an Eulerian approach, the gas phase being split in two fields. The small bubbles, assumed to be spherical, are modeled with a dispersed approach whereas the larger bubbles, considered as too distorted to be accurately described by correlations, are simulated through an interface locating method. As a first step towards this new approach, we simplify the general concept by considering the liquid phase as continuous in our simulations. The principle of this 3-field approach is summarized in Fig. 1.

* Corresponding author. Tel.: +33 (0)1 87 85 83.
*E-mail address:* romain.denefle@edf.fr (R. Denèfle).

http://dx.doi.org/10.1016/j.compfluid.2014.07.018
0045-7930/

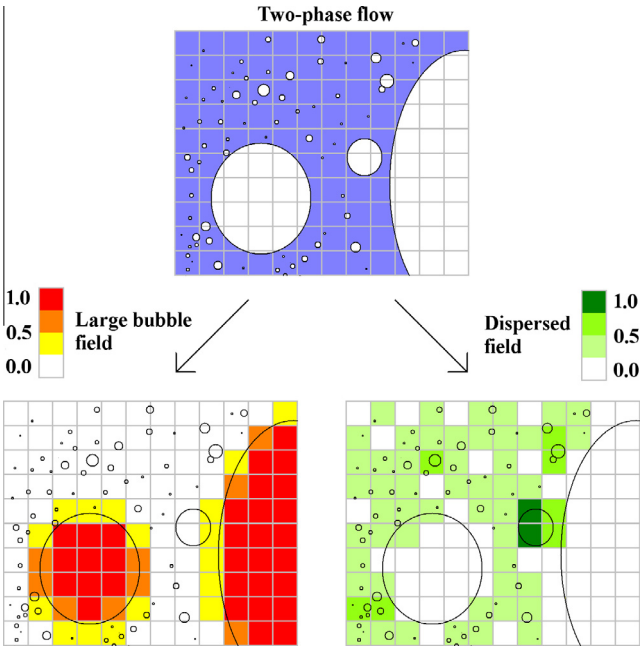


**Fig. 1.** Scheme of the principle of the multifield hybrid approach. The gas phase is split in two fields, one being dealt with a dispersed approach (green) and the other one having its interface located. (For interpretation of the references to color in this figure legend, the reader is referred to the web version of this article.)

## 2. Numerical model

These models have been implemented in the CFMD code NEPTUNE_CFD [9]. The flow motion is followed using the two-fluid model of Ishii [1] extended to $n$-phase. With the assumption of a common pressure for all fields, the system consists in three balance equations per field, describing the conservation of the mass, the momentum and the energy. The solver SIMPLE [10] is based on a finite volume discretization, together with a collocated arrangement for all variables. An iterative coupling of the equations is used to ensure both mass and energy conservation. The data structure is totally face-based, which allows the use of arbitrary-shaped cells including nonconforming meshes. Following the strategy of the code, the choice is made to keep an Eulerian point of view for every field.

In the present work, we restrict our study to adiabatic cases, simplifying the system to the mass and momentum equations for each field, respectively Eqs. (1) and (2).

$$\frac{\partial(\alpha_k\rho_k)}{\partial t} + \nabla\cdot(\alpha_k\rho_k\mathbf{U}_k) = \sum_{p\neq k}\Gamma_{p\to k} \tag{1}$$

$$\frac{\partial(\alpha_k\rho_k\mathbf{U}_k)}{\partial t} + \nabla(\alpha_k\rho_k\mathbf{U}_k\mathbf{U}_k) = -\nabla(\alpha_k P) + \alpha_k\rho_k\mathbf{g} + \nabla\cdot\left(\alpha_k\mathbb{T}_k + \alpha_k\mathbb{T}_k^{SM}\right) + \sum_{p\neq k}\mathbf{M}_{p\to k} \tag{2}$$

where $\alpha$, $\rho$, $\mathbf{U}$, $\Gamma$, $P$, $\mathbb{T}$ and $\mathbf{M}$ are respectively the volume fraction, the density, the velocity, the mass transfers, the pressure, the Reynolds tensor and the momentum transfers of the phase $k$. Because of the coupling of the two gas fields, we do have mass transfers $\Gamma$, and the interfacial momentum transfer $\mathbf{M}$ can be written as Eq. (3) where $\mathbf{U}^{Int}$ is the interfacial velocity.

$$\mathbf{M}_{p\to k} = \mathbf{M}_{p\to k}^{hydro} + \Gamma_{p\to k}\mathbf{U}_{kp}^{Int} \tag{3}$$

The modeling of the sub-grid tensor $\mathbb{T}^{SM}$ is still a major challenge for two-phase flow with located interfaces. As it is not the subject of this paper, this term is ignored assuming that a turbulence model should be implemented in future work. The closure laws for the dispersed field are the classical drag of Ishii and Zuber [11], the lift of Tomiyama et al. [12], the virtual mass of Zuber [13] and turbulent dispersion [14]. Further details can be found in Mimouni et al. [15].

$$\mathbf{M}_{p\to k}^{hydro} = \mathbf{M}_{p\to k}^{D} + \mathbf{M}_{p\to k}^{L} + \mathbf{M}_{p\to k}^{VM} + \mathbf{M}_{p\to k}^{TD} \tag{4}$$

The located gas field motion equation is closed by a local surface tension force and an internal wall law that express the relation between the liquid and the gas velocities at the interface. The surface tension Eq. (5) is implemented as a volume force through the CSF (*Continuum Surface Force*) formalism of Brackbill et al. [16].

$$\mathbf{F}_{st}^{k} = \beta_k\sigma\left[\nabla\cdot\left(\frac{\nabla c}{\|\nabla c\|}\right)\right]\nabla c dV \tag{5}$$

where $\beta_k$ is the averaged factor of the two-fluid model. Based on the volume average, the averaged factor is $\beta_k = \alpha_k$.

The internal wall law at the interface is chosen here to enforce the equality of the velocities of the two continuous fields. Therefore, the drag force Eq. (6) is implemented with a characteristic time $\tau_{drag}$ small compared to the time step ($\tau_{drag}/dt \to 0$).

$$\mathbf{F}_{D,kp} = \frac{\alpha_k\alpha_p(\mathbf{U}_k - \mathbf{U}_p)(\alpha_k\rho_k + \alpha_p\rho_p)}{\tau_{drag}} \tag{6}$$

Olsson and Kreiss [17] introduced a level-set method with an artificial compression step performed after the advection of the level-set function to ensure that the thickness of the transition layer is preserved. To ensure that the interfacial quantities such as the local curvature and the normal vector are accurately calculated, the same artificial compression step Eq. (7) is run between each physical time step. The parameters $\varepsilon$ controls this thickness, set here to a total of 5 cells ($\varepsilon = \Delta x/2$ where $\Delta x$ is the grid space scale), so that the accuracy of the calculation of interface properties such as curvature and normal vectors remains constant. The non-physical time $\tau$ is set to ensure a CFL condition lower than 1 for the compression step.

$$\frac{\partial\alpha}{\partial\tau} + \nabla\alpha(1-\alpha)\mathbf{n} = \varepsilon\nabla\cdot(\nabla\alpha) \tag{7}$$

These approaches for the simulation of either dispersed or located bubble interfaces have already been validated against experimental measurements in previous work [15,18]. As examples, the numerical predictions for the experiments of Bhaga and Weber [19] and of Brereton and Korotney [20] are presented in Fig. 2.

The two approaches are coupled through mass transfer terms between the two gas fields. Three contributions of this mass transfer term between the gas fields have been identified. The first one is the initiation of a located large bubble by accumulation and coalescence of the dispersed field. A criterion based on the dispersed phase void fraction and its gradient is settling the coefficient of this dispersed to continuous mass transfer, as presented in Eq. (8). The second is the creation of dispersed field when the located interface becomes unresolved. Pigny [21] proposed to consider a located bubble as unresolved if its diameter is lower than $8\Delta x$. As it can be seen in Fig. 3, a bubble with a diameter inferior to $8\Delta x$ will present, because of the CSF scheme for the curvature calculation, an overlap of the information used to calculate the curvature at two opposite points of the interface.

The resolution criterion for a local interface is evaluated by the comparison of the local curvature and the local void fraction gradient of the continuous gas void fraction to the space scale of the grid. Knowing the thickness and the curvature of a resolved interface, it results in a first numerical criterion for the dispersed gas creation of $\Delta x/20$, presented in Eq. (9). Finally, the interaction