International Journal of Multiphase Flow 89 (2017) 92–107

Contents lists available at ScienceDirect

International Journal of Multiphase Flow

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

Four-way coupled Eulerian–Lagrangian Direct Numerical Simulations in a vertical laminar channel flow

Jonathan Schillings[a], Olivier Doche[b,*], Mauricio Tano Retamales[c], Frédéric Bauer[b], Jonathan Deseure[a], Sedat Tardu[d]

[a] University Grenoble Alpes, LEPMI, F-38000 Grenoble, France
[b] University Grenoble Alpes, SIMaP, F-38000 Grenoble, France
[c] IN2P3 - University Grenoble Alpes, LPSC, F-38000 Grenoble, France
[d] University Grenoble Alpes, LEGI, F-38000 Grenoble, France

ARTICLE INFO

Article history:
Received 7 April 2016
Revised 6 September 2016
Accepted 22 October 2016
Available online 24 October 2016

ABSTRACT

Direct Numerical Simulations of a laminar two-phase flow into a vertical channel are investigated. An Eulerian–Lagrangian approach allows tracking each bubble position with a four-way coupling strategy, *i.e.* taking into account bubble-fluid and bubble-bubble interactions. The flow configuration has been chosen to highlight the buoyancy effects due to significant values of void fraction (high numbers of bubbles); hence the bubbles collisions and wall effects are the critical parameters to ensure the dispersion of the bubble plume. The DNS approach is self-consistent and does not rely on closure relations or empirical correlations for describing the collective bubble dynamics. It is found that the DNS predicts well the behavior of the bubble plume and its back effect on the liquid phase when compared with a mixture model and experimental data. The elastic nature of collisions, the sensitivity of the mean and RMS values of velocities and void fraction to the mesh quality are explored.

1. Introduction

Accurate simulations of two-phase flows require sophisticated numerical techniques such as Direct Numerical Simulations (DNS). DNS have originally been developed for investigating turbulent flows by reproducing and solving all the interactions between the motion scales with no regards to time averaged or space averaged operations, hence without having the need of closure models (such as the model RANS, k-ε, etc.). Huge quantitative information are obtained through DNS that can legitimately be classified as numerical experiments (Jiménez, 2003). There is a strong analogy between turbulent and dispersed two-phase flows of moderately dense suspensions (*i.e.* void fraction greater than 0.01), at least from a numerical simulation point of view. Both are three dimensional time-evolving flows and exhibit similar nonlinear complexity arising from the advection terms of the Navier–Stokes equations. In addition, perturbations of the carrier phase can be generated by the back effect of the transported dispersed phase, which could be combined (or not) with the local instability of the flow itself in the case of turbulence. The carrier phase induced perturbations can be identified and analyzed in a way similar to fluctuations occurring in a single-phase turbulent flow and analogous statistical post-treatment can be applied to the large data generated by DNS of multiphase flows. It is without saying that additional numerical and modeling complexities arise in multiphase flows coming from the interaction between phases and the choice of an appropriate coupling strategy is therefore crucial.

Various strategies for modeling two-fluid flows can be found in the literature and correspond to different levels of complexity and accuracy (*e.g.* Ishii, 1987; Zhang and Prosperetti, 1994). Yet, attempts to resolve dispersed flows by DNS arose quite recently. At this point, it is worth noting that the two-phase flow equations can be resolved in two different ways. In the first case each particle trajectory is individually monitored and calculated by means of the Newton's second law, and the carrier fluid represented by the Navier–Stokes equations is treated as a continuous phase (*i.e.* a Lagrangian–Eulerian approach, *e.g.* Boivin et al., 1998). In the second case, the two-phase flow system equations is expressed as the interaction between two continuous fluid: the dispersed and continuous phase (called two-fluid or Eulerian–Eulerian model) (Delnoij et al., 1997a; Esmaeeli and Tryggvason, 1999). The two-fluid model provides certainly the most detailed picture of the particles behavior and, since each particle geometry is fully resolved, allows a fine description of the interactions between the particles and the particles with the continuous phase. Hence,

* Corresponding author.
E-mail address: olivier.doche@phelma.grenoble-inp.fr (O. Doche).

http://dx.doi.org/10.1016/j.ijmultiphaseflow.2016.10.006
0301-9322/

this approach could be used to catch the surface deformation of bubbles, the induced perturbation of the flow field, the effect of surface tension, and/or clarify the coalescence phenomena, etc. However, tracking the interface requires a dynamic meshing around moving boundaries (Lu et al., 2006; Tryggvason et al., 2001) and since only a moderate amount of inclusions (classically a few hundred) can be computed, it cannot be yet representative of a typical industrial process. For the case of the Eulerian–Lagrangian approach, each element composing the discrete phase is treated as point-like (Laín et al., 2002; Nierhaus et al., 2007), in the sense that their geometrical properties (shapes, size) are implicitly taken into account in the formulation of the interfacial forces. During the last two decades, these models have widely been used to investigate the effects of particle transport on the turbulent field characteristics (*e.g.* Berg et al., 2006; Mazzitelli et al., 2003). Among those works, one can distinguish between studies neglecting the discrete phase effects on the flow (*i.e.* one-way coupling), and approaches taking into account their back reactions *via* a dedicated term expressed in the momentum equation of the continuous phase (*i.e.* two-way coupling strategy) (Druzhinin and Elghobashi, 2001; Squires and Eaton, 1990). However, it is clear that for the moment none of these DNS-like resolutions can be applied in a generic industrial context.

Bubbly flows are found in several transformation processes (bubble columns for phase extractions, electrolysis processes for hydrogen production, etc.) and they turned to be a classical subject for various academic studies (*e.g.* Akita and Yoshida, 1974; Lapin and Lübbert, 1994). In the singular case of electrochemical conversion which is one of the mostly employed process in metal production (*e.g.* aluminum) or in gas production (*e.g.* chlorine), bubbly flows have drastic effects on the overall efficiency of electrochemical cells. From the modeler point of view, it is clear that the most challenging point is to identify a way to predict accurately the various terms arising from the collective effects of bubbles (*e.g.* migration, shear-induced diffusion, etc.) applying to a mixture model (*i.e.* based on spatial and temporal averages of both phases). To the authors knowledge, these closure terms can be derived only from a few and dedicated experimental data (Leighton and Acrivos, 1987; Nicolai et al., 1995, etc.). Hence, without any reliable stand-alone formulation for the collective effect modeling, only qualitative results can be gained from two-phase flows numerical approaches as presented by Dahlkild (2001) or Schillings et al. (2015).

In the present paper, we report the study of a bubbly flow in a vertical channel by means of an Eulerian–Lagrangian DNS formulation which allows a standalone numerical resolution of the whole problem (*i.e.* without "external" information). The two-phase flow resolution and its coupling methodology are detailed and applied to a vertical channel where the amount of gaseous product generated continuously along the walls (*e.g.* electrodes) leads to a significant effect on the continuous phase velocity. The main objective of the paper is to take into account the four-way coupling which is essential in such configuration because no wall-normal bubble plume dispersion can occur without bubble-bubble interactions. A typical electrolysis cell for hydrogen production is taken as the reference simulation case and the DNS results are confronted to numerical and experimental results from the literature. The Eulerian–Lagrangian model is further employed to explore the elastic nature of collisions, the sensitivity of the mean and RMS values of velocities and void fraction to the mesh quality or the local values of wall shear-stress. All of the simulations presented in this work are performed for established laminar inlet flows.

2. Numerical model

In this paper, the dynamic of a two-phase bubbly flow is investigated in a vertical channel through a straight Eulerian–Lagrangian formulation. The electrolyte motion is described by the well-known time dependent and three-dimensional Navier–Stokes equations under assumptions of a Newtonian and incompressible liquid as the continuous phase. The isothermal condition is also assumed for the whole domain since buoyancy effect due to temperature is negligible in comparison to the two-phase buoyancy under the present operating conditions. Gas bubbles trajectories are computed by resolving the Newton's second law for each of them. As stated before, the resolution is performed with a four-way coupling (*i.e.* accounting bubble-bubble and bubble-liquid interactions), including the effect of the wall on the bubbles trajectories. The configuration used throughout this work is depicted in Fig. 1. The gas-evolving lateral channel walls are set vertically in order to generate buoyancy forces in the streamwise direction, $\boldsymbol{x}$. They are located at $y = 0$ and $y = 2h$, start at $x = 0$ and end at $x = L_x$ and their size along the $\boldsymbol{z}$ direction is considered as infinite in order to simulate pseudo-2D cases. In this work, the liquid phase is restricted to the laminar regime.

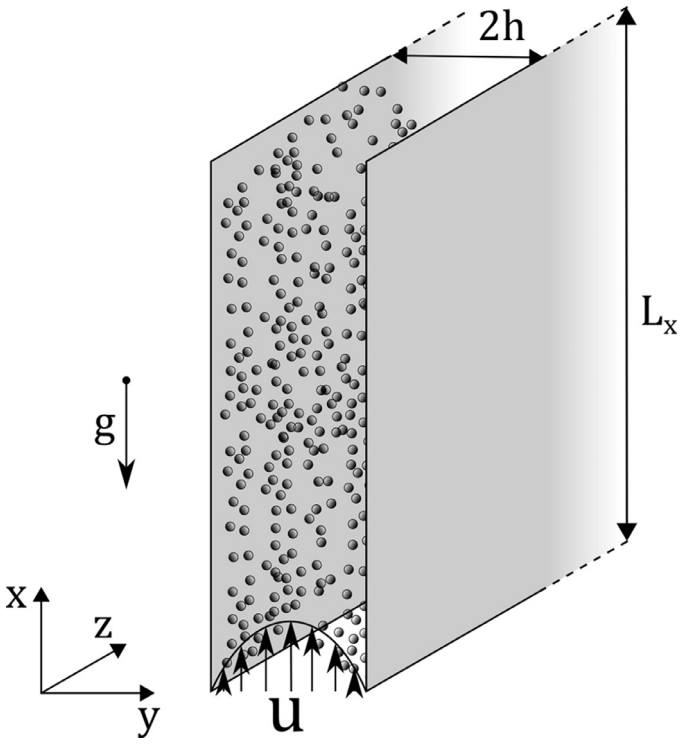

Fig. 1. Scheme of the electrodes and flow configuration.

Numerical simulations are performed with a validated finite difference code developed by our team (Bauer et al., 2015; Doche et al., 2013, 2012). This code, optimized for turbulent simulations, handles massive parallel computing and fits the requirement for strong flow couplings with large number of inclusions.

The DNS data are compared to the simulations of a homogenous (also called *mixture*) model here. This model considers the gas-liquid mixture as a unique continuous Newtonian fluid governed by momentum and volume conservation equations. The equations depend on the void fraction which is a scalar variable representing the local proportion of dispersed phase in the mixture volume. The evolution of the void fraction is resolved by means of the *relative flux*, a phenomenological closure term extracted from theoretical and empirical studies that specifies the superficial velocity of the dispersed phase relatively to the mixture motion. The complete mixture model formulation can be found in our previous work (Schillings et al., 2015).

The following subsections detail the Eulerian–Lagrangian numerical models resolved for each phase and their interactions.