Chemical Engineering Science 109 (2014) 65–74

Contents lists available at ScienceDirect

Chemical Engineering Science

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

A critical comparison of surface tension models for the volume of fluid method

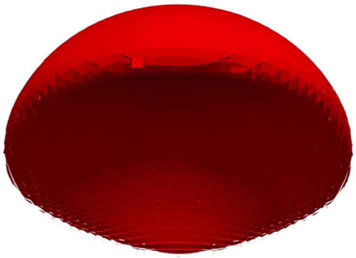
M.W. Baltussen, J.A.M. Kuipers, N.G. Deen *

Multiphase Reactors Group, Department of Chemical Engineering and Chemistry, Eindhoven University of Technology, P.O. Box 513, 5612 AZ Eindhoven, The Netherlands

HIGHLIGHTS

- Three different surface tension models for VOF are compared.
- Simulations were done for a wide range of Eotvos (Eo) and Morton (Mo) numbers.
- The height function model performs best for Eo < 1.
- The tensile force model performs best for high Eo < 10.

GRAPHICAL ABSTRACT



ARTICLE INFO

Article history:
Received 16 August 2013
Received in revised form
5 December 2013
Accepted 31 December 2013
Available online 31 January 2014

Keywords:
Multiphase flow
Multiscale
Volume of fluid model
Bubble columns
Hydrodynamics
Surface tension modeling

ABSTRACT

In many different fields of research, the interactions between two immiscible fluids are of importance. To study these flows in industrial equipment, a multi-scale modeling approach is often used. In this approach, the smallest scale models apply detailed information in the form of closure equations for the larger scale models, which can model complete industrial equipment. This paper will focus on the improvement of the smallest scale model; direct numerical simulations employing the Volume of Fluid model. In this model, mass is inherently conserved because of the surface treatment, but this treatment also poses a challenge in calculating the surface properties like the surface tension. In this paper, three different surface tension models for the Volume of Fluid were tested: the generally used Continuum Surface Force (CSF) model, the height function model and the novel tensile force method. From the verification tests, it was concluded that both the height function model and the tensile force method are an improvement of the CSF model. The single bubble simulations showed that the height function method works best for small bubble ($Eo < 1$). This is due to problems with connectivity for the tensile force method. While for the larger bubbles ($Eo > 10$), the tensile force method is the best functioning surface tension model, because the calculation of the curvature in the height function method uses a stencil in which the distance between two interfaces in the direction of the normal should at least be four grid cells. In all the other tested cases, the height function model and the tensile force method perform equally well. The Morton number changes the ranges for the region of use of the surface tension models slightly when $\log Mo \leq -7$ (the height function model can only be used when $Eo \leq 2$, while the tensile force method can only be used at $Eo \geq 2$) and $\log Mo \geq 1$ (the height function model can in this region also be used when $Eo > 10$).

1. Introduction

Clouds, fuel injection and bubble columns are a few examples from geophysics, environmental studies, engineering practice and fundamental physics in which the interactions between two immiscible fluids are of importance (Scardovelli and Zaleski, 1999; Kwakkel

---

* Corresponding author. Tel.: +31 40 247 3681; fax: +31 40 247 5833.
E-mail address: N.G.Deen@tue.nl (N.G. Deen).

0009-2509/$ - see front matter © 2014 Elsevier Ltd. All rights reserved.
http://dx.doi.org/10.1016/j.ces.2013.12.045

et al., 2012). These interactions can be studied using Computational Fluid Dynamics (CFD). However, CFD models are not able to resolve all details of the bubbly flow in for example an industrial size bubble column. To solve this issue, a multi-scale modeling approach can be used to accurately describe the hydrodynamics in these large bubble columns (Raessi et al., 2010; Roghair et al., 2011).

In the multi-scale modeling strategy (Fig. 1), there are three kinds of models: Euler–Euler models, Euler–Lagrange models and Direct Numerical Simulations (DNS). Euler–Euler models describe industrial size bubble columns by assuming both the liquid and the bubble phase as interpenetrating liquids. However, Euler–Euler models need closure relations for the bubble–bubble and bubble–liquid interactions (Yang et al., 2007; Roghair et al., 2011).

These closure relations can partly be obtained using Euler–Lagrange models, which are intermediate scale models. In these models, the flow field is determined with the Navier–Stokes equations, while the bubbles are individually described as Lagrangian spheres. Because not all the bubble dynamics are resolved in detail, the model still needs closures for the interaction forces between the phases, such as the lift force and the drag force (Yang et al., 2007; Roghair et al., 2011).

The smallest scale model of the multi-scale multiphase modeling approach is DNS. DNS is a fully resolved simulation method, i.e. the Navier–Stokes equations are solved without a priori assumptions. Although DNS methods are able to very accurately describe bubbly flows, these simulations can only be performed for $O(10^2)$ bubbles because of the high computational costs.

There are many different DNS methods available in the literature for the modeling of two immiscible fluids. These models can be divided into two groups: fixed grid methods and moving grid methods. In moving grid methods, the grid is aligned with the interface and will thus create two sub domains, while in fixed grid methods a stationary (Eulerian) grid is used. The advantage of fixed grid methods is that they are able to handle strong topological changes. However, the calculation of the surface properties such as the surface tension is challenging, because the exact location of the interface is unknown (Scardovelli and Zaleski, 1999; van Sint Annaland et al., 2005; Roghair et al., 2011). In this study highly deformed bubbles will be studied, consequently a fixed grid method is chosen.

The main fixed grid methods are Front Tracking (FT), Level-Set (LS) and Volume Of Fluid (VOF) (Son, 2003; Albadawi et al., 2013). The substantial difference between these methods is the manner of the interface treatment. Only the FT model directly tracks the surface with a Lagrangian grid. Therefore, the position and shape of the interface are known, which enables an easy and accurate description of the interface physics and it avoids the use of a highly refined grid and numerical diffusion of the interface. However, there are three major disadvantages of the FT model. First of all, the tracking of the surface is complex especially if merging and break-up of the bubbles are considered. In addition, due to topological changes of the interface, restructuring of the Lagrangian marker particles is necessary for an accurate description. Finally, mass conservation is a problem in the model, due to the mapping of information from the Eulerian grid to the Lagrangian grid and vice versa (Chang et al., 1996; Shin and Juric, 2002; van Sint Annaland et al., 2005; Kwakkel et al., 2012).

Both the VOF and the LS do not directly track the interface, therefore both methods are front capturing techniques. The LS captures the interface with the Level-Set function (F). This distance function is zero at the interface, negative in one fluid and positive in the other fluid. The interface is advected by the local fluid velocity according to the following equation:

$$\frac{DF}{Dt} = \frac{\partial F}{\partial t} + \mathbf{u} \cdot \nabla F = 0 \tag{1}$$

The advantage of the usage of a smooth distance function is the accurate description of the surface and thereby an accurate

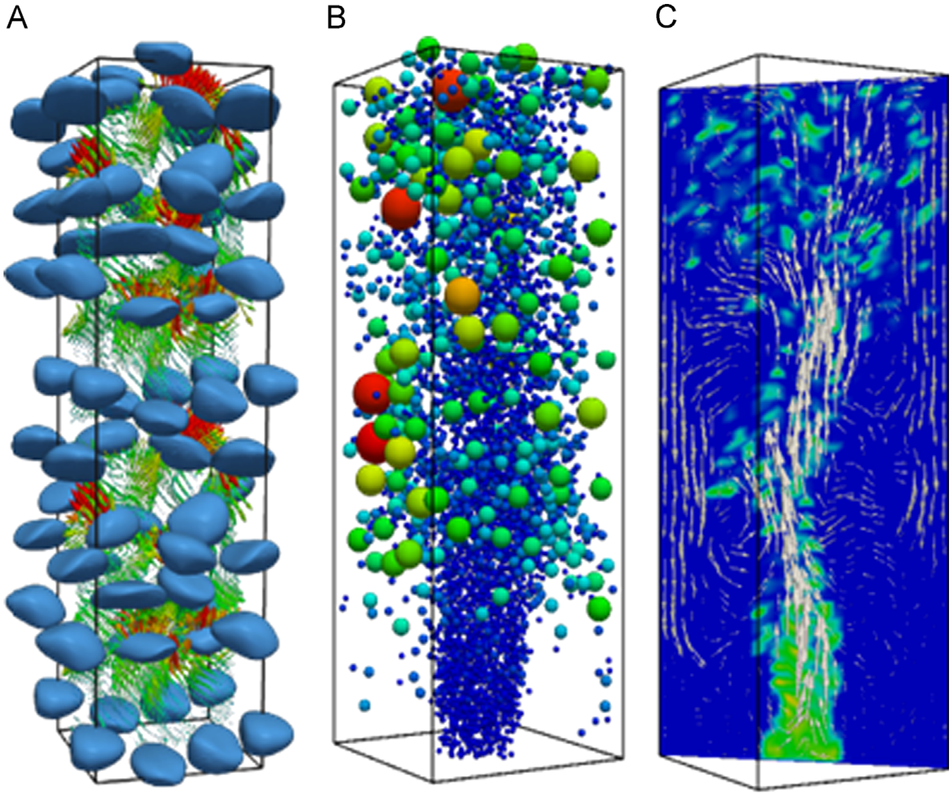


Fig. 1. The multi-scale modeling approach for gas-liquid flow (Roghair et al., 2011). Figure A shows a simulation using Direct Numerical Simulations, Figure B a simulation with the Euler–Lagrange model and Figure C a simulation with the Euler–Euler model.