DIRECT NUMERICAL SIMULATION OF PARTICLE MIXING IN DISPERSED GAS-LIQUID-SOLID FLOWS USING A COMBINED VOLUME OF FLUID AND DISCRETE PARTICLE APPROACH

Niels G. DEEN, Martin VAN SINT ANNALAND and J.A.M. KUIPERS

University of Twente, Fac. of Science and Technology, PO Box 217, 7500 AE Enschede, THE NETHERLANDS

ABSTRACT
In this paper a hybrid model is presented for the numerical simulation of gas-liquid-solid flows using a combined Volume Of Fluid (VOF) and Discrete Particle (DP) approach applied for respectively dispersed gas bubbles and solid particles present in the continuous liquid phase. The hard sphere DP model, originally developed by Hoomans et al. (1996) for dense gas-solid systems, has been extended to account for all additional forces acting on particles suspended in a viscous liquid and has been combined with the VOF model presented recently by van Sint Annaland et al. (2005) for complex free surface flows. In this paper the physical foundation of the combined VOF-DP model will be presented together with illustrative computational results highlighting the capabilities of this hybrid model. The effect of bubble-induced particle mixing has been studied focusing on the effect of the volumetric particle concentration. In addition particle mixing was studied in systems with coalescing bubbles.

NOMENCLATURE

\( a \) Computational domain in x-direction (m)
\( C_d \) Drag coeff. for an isolated spherical particle (-)
\( C_L \) Lift coefficient (-)
\( C_{VM} \) Virtual mass coefficient (-)
\( d \) Plane constant for interface segment cutting through Eulerian cell (-)
\( d_e \) Equivalent bubble diameter (m)
\( d_p \) Particle diameter (-)
\( D \) Distribution or smoothening function (-)
\( E_o \) Eötvös number (-)
\( F \) Fractional amount of liquid (-)
\( \tilde{F} \) Smoothed colour function (-)
\( F_s \) Volumetric surface tension force (N/m\(^3\))
\( h \) Width of the computational stencil for the smoothening function (m)
\( M \) Morton number (-)
\( m_p \) Particle mass (kg)
\( N_p \) Number of suspended solid particles (-)
\( n_i \) \( i^{th} \) component of the unit normal vector (-)
\( p \) Pressure (N/m\(^2\))
\( R \) Bubble radius (m)
\( Re \) Reynolds number (-)
\( t \) Time (s)
\( U \) Superficial velocity (m/s)

Greek letters
\( \beta_p \) Effective fluid-particle friction coefficient (kg/(m\(^3\).s))
\( \phi \) Dimensionless liquid velocity (-)
\( \epsilon_i \) Liquid phase volume fraction (-)
\( \kappa \) Curvature (m\(^{-1}\))
\( \mu \) Dynamic viscosity (kg/(m.s))
\( \rho \) Density (kg/m\(^3\))
\( \Delta \rho \) Density difference (kg/m\(^3\))
\( \Delta z \) Grid spacing in \( i^{th} \) co-ordinate direction (m)
\( \sigma \) Surface tension (N/m)
\( \xi_i \) Dimensionless \( i^{th} \) co-ordinate direction

Vectors
\( \vec{F} \) Gravity force (N)
\( \vec{F}_p \) Pressure force (N)
\( \vec{F}_d \) Drag force (N)
\( \vec{F}_L \) Lift force (N)
\( \vec{F}_{VM} \) Virtual mass force (N)
\( \vec{F}_s \) Volumetric surface tension force (N/m\(^3\))

\( F_{s,\text{fs}} \) Volumetric force due to fluid-solid coupling (N/m\(^3\))
\( g \) Gravitational acceleration (m/s\(^2\))
\( m \) Normal vector
\( n \) Unit normal vector
\( \nu \) Fluid velocity (m/s)
\( \nu \) Particle velocity (m/s)

Subscripts and superscripts
\( g \) Gas phase
\( l \) Liquid phase
\( p \) Particle phase
\( x \) x-direction
\( y \) y-direction
\( z \) z-direction
INTRODUCTION

Multiphase systems are frequently encountered in a variety of industrial processes involving a.o. coating, granulation, drying and synthesis of fuels (Fischer Tropsch) and base chemicals. The hydrodynamics of multiphase systems is dictated by the motion of the individual phases and the complex mutual interactions and as a direct consequence thereof CFD-based modelling of these systems has proven notoriously difficult. In literature both the Eulerian approach (Torvik and Svendsen, 1990; Mitra-Majumdar et al., 1997) and Lagrangian approach (Bourloutski and Sommerfeld, 2002) have been adopted to study gas-liquid-solid three-phase flows. Although these computational methods are in principle well-suited to simulate the large-scale flow behaviour of slurry bubble columns and three-phase fluidized beds (i.e. the time-averaged circulation pattern and spatial gas holdup distribution), problems arise related to the representation of the interactions between the individual phases leading to considerable closure problems. To overcome this problem detailed microscopic models can be used to generate closure laws which are needed in coarse-grained simulation models which are used to describe the macroscopic behaviour and as a practical consequence thereof do not resolve all the relevant length and time scales. This multi-scale modelling approach has been used by the authors previously for dense gas-solid flows (van der Hoef et al., 2004) and for dispersed gas-liquid two-phase flows (van Sint Annaland et al., 2003 and Deen et al., 2004a, 2004b) and can in principle be used for other types of multiphase flow as well. Our model is based on a combined Volume Of Fluid (VOF) and Discrete Particle (DP) approach applied for respectively dispersed gas bubbles and solid particles present in the continuous liquid phase.

Volume Of Fluid (VOF) methods (Hirt and Nichols, 1981; Youngs, 1982; Rudman, 1997, 1998; Rider and Kothe, 1998; Scardovelli and Zaleski, 1999; Popinet and Zaleski, 1999; Bussman et al., 1999) employ a colour function \( \Phi(x,y,z,t) \) that indicates the fractional amount of fluid present at a certain position \((x,y,z)\) at time \(t\). The evolution equation for \( \Phi \) is usually solved using special advection schemes (such as geometrical advection, a pseudo-Lagrangian technique), in order to minimize numerical diffusion. In addition to the value of the colour function the interface orientation needs to be determined, which follows from the gradient of the colour function. Roughly two important classes of VOF methods can be distinguished with respect to the representation of the interface, namely Simple Line Interface Calculation (SLIC) and Piecewise Linear Interface Calculation (PLIC). Earlier work is generally typified by the SLIC algorithm due to Noh and Woodward (1976) and the Donor-Acceptor algorithm published by Hirt and Nichols (1981). Modern VOF techniques include the PLIC method due to Youngs (1982). The accuracy and capabilities of the modern PLIC VOF algorithms greatly exceeds that of the older VOF algorithms such as the Hirt and Nichols VOF method (Rudman, 1997). A drawback of VOF methods is the so-called artificial (or numerical) coalescence of gas bubbles which occurs when their mutual distances is less than the size of the computational cell.

Discrete Particles (DP) methods employ particle tracking taking into account i) all relevant external forces (gravity, pressure, drag, lift and virtual mass forces) acting on the particles and ii) collisions between particles and confining walls. Finally, two-way coupling, which becomes important at high solids volume fraction, is also taken into account. In this study the hard sphere DP model, originally developed by Hoomans et al. (1996) for gas-solid systems (also see Hoomans, 2000), has been extended to account for all additional forces acting on particles suspended in a viscous liquid and has been combined with the VOF model presented recently by van Sint Annaland et al. (2005). The direct numerical simulation has been pioneered by Fan and co-workers (Li et al., 1999, 2000, 2001; Zhang et al., 2000a, 2000b) who used a two-dimensional model based on the VOF method to track the bubble interface.

The organisation of this paper is as follows: first the description of the model is given followed by a description of the numerical solution method. Subsequently the results are presented and discussed and finally the conclusions are presented.

MODEL DESCRIPTION

Our model consists of two major sub-models, the model dealing with the motion of the gas bubbles in the continuous phase (consisting of liquid and suspended solid particles) and the DP model dealing with the motion of the suspended particles taking into account the action of external forces and non-ideal particle-particle and particle-wall collisions. First the VOF model is briefly described followed by a brief description of the DP model.

Volume of fluid model

The governing conservation equations for unsteady, Newtonian, multi-fluid flows are given by the following expressions:

\[
\frac{\partial \varepsilon}{\partial t} + \nabla \cdot (\varepsilon \mathbf{u}) = 0
\]

\[
\rho \varepsilon \frac{\partial}{\partial t}(\varepsilon \mathbf{u}) + (\varepsilon \nabla \cdot \mathbf{u}) = -\varepsilon \nabla p + \rho \mathbf{u} - \mathbf{F}_{\varepsilon-\varepsilon'} + \mathbf{F}_{\varepsilon}\]

The source terms \( \mathbf{F}_{\varepsilon} \) and \( \mathbf{F}_{\varepsilon-\varepsilon'} \) account respectively for the forces due to surface tension and two-way coupling due to the presence of the suspended solid particles. Furthermore \( \varepsilon \) represents the volume fraction of the continuous liquid phase. The mixture density \( \rho \) and viscosity \( \mu \) are evaluated from the local distribution of the phase indicator or colour function function \( F \) which is governed for by:

\[
\rho = \rho_c (1 - \varepsilon) + \rho_s \varepsilon
\]

\[
\mu = \mu_c (1 - \varepsilon) + \mu_s \varepsilon
\]
expressing that the interface property is advected with the local fluid velocity. For the local average density $\rho$ linear weighing of the densities of the liquid (l) and gas phase (g) is used:

$$\rho = F \rho_l + (1-F) \rho_g$$  \hspace{1cm} (4)

Similarly, the local average dynamic viscosity can also be obtained via linear averaging of the dynamic viscosities of the liquid and gas phase. As an alternative, a more fundamental approach recently proposed by Prosperetti (2001), the local average viscosity can be calculated via harmonic averaging of the kinematic viscosities of the involved phases according to the following expression:

$$\mu = \frac{F \mu_l + (1-F) \mu_g}{\mu_l + (1-F) \mu_g}$$  \hspace{1cm} (5)

In all computations reported in this paper Eq. (5) was used to compute the local average viscosity. The volumetric surface tension force appearing in the momentum Eq. (2) acts only in the vicinity of the interface.

Discrete particle model

The motion of the suspended solid particles is given by the Newton’s second law given by the following expression which accounts for the action of gravity, far-field pressure, drag, lift and added mass forces.

$$\frac{d}{dt} \left( m \frac{\mathbf{v}}{\mathbf{r}} \right) = \mathbf{F}_\mathbf{d} + \mathbf{F}_\mathbf{l} + \mathbf{F}_\mathbf{v} + \mathbf{F}_\mathbf{a}$$

$$= m \mathbf{g}_\mathbf{f} - \rho v \mathbf{F}_\mathbf{p} + \frac{\rho}{(1-\epsilon_f)} \frac{V_f}{C} \left( \mathbf{v} - \mathbf{v}_\mathbf{f} \right)$$

$$- C_f \rho \mathbf{F}_\mathbf{p} \left( \mathbf{v} - \mathbf{v}_\mathbf{f} \right) \times (\mathbf{v} \times \mathbf{v}_\mathbf{f})$$

$$- \rho V_f C_f \frac{D t}{D t} \left( \mathbf{v} - \mathbf{v}_\mathbf{f} \right) + (\mathbf{v} - \mathbf{v}_\mathbf{f}) \cdot (\nabla \mathbf{v})$$

This equation is similar to the one used by Delnoij et al. (1997) in their discrete bubble model for dispersed gas-liquid two-phase flow. Since the size of the particles typically is (much) smaller than the size of the computational grid required for capturing the bubble dynamics, a closure equation is required for the effective drag (friction) acting on the particles. Following Hoomans et al. (1996) the friction between the continuous liquid phase and the suspended solid particles is given by the Ergun equation in the dense regime ($\epsilon_f < 0.8$) and the Wen and Yu equation in the dilute regime ($\epsilon_f > 0.8$) respectively given by ($\epsilon_f < 0.8$):

$$\beta_f = \frac{150 (1-\epsilon_f)^2}{\epsilon_f (K^2)} + 1.75 (1-\epsilon_f) \frac{\rho_l}{d_p} \left| \mathbf{v} - \mathbf{v}_\mathbf{f} \right|$$  \hspace{1cm} (7a)

and ($\epsilon_f > 0.8$):

$$\beta_f = \frac{3}{4} C_w \frac{\epsilon_f (1-\epsilon_f)}{d_p} \left[ \mathbf{v} - \mathbf{v}_\mathbf{f} \right]$$  \hspace{1cm} (7b)

where the drag coefficient $C_w$ follows well-known equations for drag coefficients for spheres whereas $C_{L_M} = 0.5$ and $C_L = 0.5$ was assumed. Alternatively drag closure equations obtained from LB models can be used for the effective drag closure (van der Hoef et al., 2004, 2005). For the (possible) non-ideal particle-particle and particle-wall collisions we use a three parameter model accounting for normal and tangential restitution and friction. The associated collision parameters will respectively be denoted by $e_n$, $e_t$ and $\mu$m.

NUMERICAL SOLUTION METHOD

Computation of the flow field

The Navier-Stokes equations can be solved with a standard finite volume technique on a staggered rectangular three-dimensional grid using a two-step projection-correction method with an implicit treatment of the pressure gradient and explicit treatment of the convection and diffusion terms. A second order flux delimited Barton-scheme (Centrella and Wilson, 1984) is used for the discretisation of the convection terms and standard second order central finite differences for the diffusion terms. We use a robust and very efficient Incomplete Cholesky Conjugate Gradient (ICCG) algorithm to solve the Pressure Poisson Equation (PPE).

Computation of the surface force

In the CSF model (Brackbill et al., 1992) the surface tension force acts via a source term $\tau_s$ in the momentum equation which only acts in the vicinity of the interface. The expression for $\tau_s$ is given by

$$\tau_s = 2 \sigma \kappa \mathbf{m}$$  \hspace{1cm} (8)

where the expression for the curvature is obtained from the divergence of the unit normal vector to the interface:

$$\kappa = \left( \nabla \cdot \mathbf{n} \right) = \frac{1}{\left| \mathbf{n} \right|} \left( \mathbf{n} \cdot \nabla \left| \mathbf{n} \right| - \left( \nabla \cdot \mathbf{n} \right) \right)$$  \hspace{1cm} (9)

The normal to the interface is computed from the gradient of the smoothed colour function. The smoothing technique used in this paper will be discussed later.

Solution of the F-advection equation

The integration of the hyperbolic F-advection equation is the most critical part of the VOF model and is based on geometrical advection which can be viewed as a pseudo-Lagrangian advection step. The advantage of the geometrical advection is given by the fact that a very sharp interface is maintained during the simulations. First for each Eulerian cell containing an interface the unit normal vector to the interface is estimated from the gradient of the colour function $F$:

$$\mathbf{n} = \frac{\nabla \mathbf{F}}{\left| \nabla \mathbf{F} \right|}$$  \hspace{1cm} (10)

where the subscript refers to the co-ordinate direction. These transformations involve i) change of co-ordinate directions ii) mutual interchange of co-ordinate directions and iii) interchange of the dispersed and continuous phase. Through the use of these transformations it is possible to reduce the total number of possible interface configurations from sixty four to five generic ones which are schematically shown in Fig. 1. From these five generic interface types the particular type prevailing in a certain Eulerian cell needs to be determined on basis of the known interface orientation (i.e. the normal vector to the interface) and the F-value of the interface cell. For the computation of the fluxes through the cell faces the equation for the planar interface segment cutting through the Eulerian cell needs to be considered. This equation is given by:

$$n_i n_i \xi + n_j n_j \eta + n_k n_k \zeta = d$$  \hspace{1cm} (12)

where $\xi$ ($i = 1..3$) represents the dimensionless co-ordinate in direction $i$ given by:

$$\xi_i = \frac{x_i}{\Delta x_i}$$  \hspace{1cm} (13)

where $\Delta x_i$ represents the grid-spacing in co-ordinate direction $x_i$ ($i = 1..3$). The value of the plane constant $d$ can be determined by equating the expression for the dimensionless liquid volume (volume below the planar interface segments shown in Fig. 1) to the known fractional amount of liquid or the F-value in the interface cell. The value of $d$ can be obtained readily from the root of these non-linear equations using the Newton-Raphson method which needs however to be done with care in order to find the correct root of the cubic equations. As an alternative the Regula Falsi method can be used, which requires however an interval in which the root can be found. This interval can be obtained on basis of the known interface orientation (i.e. components of the normal to the interface) and the fractional amount of liquid in the interface cell (i.e. the F-value) using simple geometrical considerations. One should keep in mind here that the solution of the non-linear equation needs to be carried out only for the interface cells.

Once the aforementioned steps have been taken, finally the amount of liquid fluxed through each of the faces of the Eulerian cells during a time step $\Delta t$ can be computed. The F-advection equation is discretised with an explicit treatment of the convections terms, where a straightforward generalisation of the 2D geometrical advection method given by Delnoij (1999) is used (also see Scardovelli and Zaleski, 1999). In our implementation of this method we have adopted the split advection scheme. Because the expressions for the fluxes through the cell faces are quite lengthy they are not given here. Finally the computed new F-values are corrected for (small) non-zero divergence of the velocity field due to the iterative solution of the Pressure Poisson Equation (PPE).

**Smoothening of the colour function F**

As indicated before the interface orientation (i.e. the normal to the interface) is computed from the gradient of the color function $F$ according to Eq. (8). Basically this involves numerical differentiation of a discontinuous function leading in practice to (small) inaccuracies. This problem can be overcome however by making use of a smooth color function $\tilde{F}$ for the computation of the unit normal to the interface using Eq. (8) with $F$ replaced by $\tilde{F}$ obtained from:

$$\tilde{F}(\widetilde{\mathbf{x}}) = \sum D(x-x_i)D(y-y_i)D(z-z_i)F(\widetilde{\mathbf{x}})$$  \hspace{1cm} (14)

where the smoothening function $D$ is given by the function proposed by Peskin (1977):

$$D(x) = \frac{1}{2h} (1 + \cos(\pi x/h))$$  \hspace{1cm} (15)

or as an alternative by a suitable polynomial expression as the one proposed by Deen et al. (2004b):

$$D(x) = \frac{15}{16} \left( \frac{x}{h} \right)^2 - \frac{2}{3} \frac{x}{h} + 1$$  \hspace{1cm} (16)

where $h$ represents the width of the computational stencil used for the smoothening. The summation in Eq. (14) only involves the grid points with distance (in each separate co-ordinate direction) equal or less then the smoothening or filter width $h$. We typically use $h = 2\Delta x$ where $\Delta x$ represents the Eulerian grid size and, unless otherwise stated. The width of the computational stencil for the smoothening should be selected carefully. When the width is too small numerical instabilities may arise, especially in case the coefficient of surface tension is high. On the other hand when the width of the computational stencil is chosen too large, excessive smoothening (“thickening” of the interface) is obtained which is undesirable. For the simulations reported in this paper we used Eq. (16) and additionally we used the smooth colour function $\tilde{F}$ instead of $F$ in Eq. (8). It should be stressed here that this smoothed colour function is only used in conjunction with the estimation of the unit normal to the interface and not in the computation of the material fluxes through the faces.
of the computational cells for which the unsmoothed colour function was used.

**Computation of the particle motion and particle-fluid coupling**

The equation of motion for the particles is solved with a simple first order integration scheme with linear implicit treatment of the effective drag and added mass forces. Where required the Eulerian quantities such as the liquid phase volume fraction, pressure and velocity components as well as the spatial derivatives of these quantities are interpolated from the staggered Eulerian grid using volume-weighing. The non-ideal particle-particle and particle-wall collisions are taken into account with an event-driven computational scheme using efficient Molecular Dynamics (MD) type of schemes (see Hoomans et al., 1996 and Hoomans, 2000). In the present version of the model we have not accounted for changes in the collision dynamics in case the particles are very close to each other (see Li et al., 1999). The particle-fluid coupling requires Euler-Lagrange and Lagrange-Euler mapping for which we use a volume-weighing technique.

**RESULTS**

Prior to the presentation of the main results the verification of our computational model will be briefly discussed which involves tests for the flow solver and the two-way coupling of the dispersed phases. For a detailed test of the encounter model we refer to Hoomans et al. (1996) and Hoomans (2000).

**Model verification**

In our first test the correctness of the implementation of the flow solver was tested for fully developed laminar flow in a square channel with diameter 2a. For this particular case the expression for the fully developed velocity profile can be derived analytically and is given by:

$$v = \frac{a^2 \Delta \rho}{2 \mu L} \left(1 - \frac{x}{a}\right)$$

where $x$ is the $x$-coordinate corresponding to the main flow direction. For the simulation the density and viscosity of the fluid were set to respectively 1000 kg/m$^3$ and 1.0 kg/(m.s) whereas the $x$-velocity at the inlet was set to 1.0 m/s. In Fig. 3 the dimensionless velocity profiles are shown for the central plane cutting through a square duct with diameter 0.02 m obtained from the analytical solution (+) and the numerical solution (x) using 40 computational cells in both the $x$- and $y$-direction and 100 cells in the flow direction (duct length 1.0 m). In Fig. 3 the dimensionless velocity profiles are shown for the central plane cutting through a square duct with diameter 0.02 m obtained from the analytical solution (+) and the numerical solution (x) using 40 computational cells in both the $x$- and $y$-direction and 100 cells in the flow direction (duct length 1.0 m). Whereas the inlet and outlet respectively prescribed (uniform) inflow and prescribed pressure boundary conditions were taken.

In our second test the correctness of the implementation of two-way coupling for the particles was tested by comparing the computed pressure gradient for liquid flow through a fixed bed of particles with the pressure gradient according to the Ergun equation:

$$-\frac{dp}{dz} = 150 \left(1 - \varepsilon_0\right)^2 \frac{\mu U_m}{\varepsilon_0^2 d_p} + 1.75 \frac{\mu}{\varepsilon_0 d_p} \Omega U_m^2$$  \hspace{1cm} (18)$$

In this test the liquid phase flow field and associated pressure drop was computed for a stationary array of 25x25x125 particles with a diameter of 3.95 mm packed in a square column with a diameter of 0.1 m and a length of 1.0 m. Free-slip boundary conditions were used for the walls of the duct whereas for the inlet and outlet respectively prescribed (uniform) inflow and prescribed pressure boundary conditions were taken. In this case the bed porosity equals 0.495792. The density and viscosity of the fluid were set to respectively 1000 kg/m$^3$ and 0.01 kg/(m.s) whereas the superficial velocity at the inlet was set to 0.1 m/s. For these conditions the pressure gradient according to the Ergun equation equals 20,335 Pa/m.

From the numerical model, using 10 computational cells in both the $x$- and $y$-direction and 50 cells in the flow direction, an average pressure gradient of 19,273 Pa/m was obtained leading to a relative error of 5.2%. This error can be reduced further by using more computational cells.

Finally van Sint Annaland et al. (2005) performed extensive calculations using their VOF-model for gas bubbles rising in quiescent viscous liquids and demonstrated that the computed terminal rise velocities and shapes of the bubbles agreed very well with those obtained from the Grace diagram (see next section) over a very wide range of Eötvös and Morton numbers, while using a high density and viscosity ratio characteristic for gas-liquid systems. They also showed that i) the size of
the computational domain in the lateral directions should

equal 3 to 4 times the diameter of the bubble to reduce the wall effect to an acceptable level ii) for the gas bubble typically 16 to 20 computational cells (in each direction) are required to correctly predict the bubble characteristics (rise velocity and shape). Van Sint Annaland et al. (2005) also successfully computed the co-axial and oblique coalescence of two bubbles in a viscous liquid, a process which has been studied experimentally by Brereton and Korotney (1991). Clearly the agreement between the computed results and the results obtained from the analytical expressions is very good and therefore our model can be applied with confidence to more complex problems such as gas-liquid-solid systems.

**Single gas bubbles**

Grace (1973) has analysed a large body of experimental data on shapes and rise velocities of bubbles in quiescent viscous liquids and has shown that this data can be condensed into one diagram, provided that an appropriate set of dimensionless numbers is used. Although this diagram is strictly not valid for gas-liquid-solid systems it is nevertheless useful for reference purposes. A copy of this diagram, taken from Clift et al. (1978) is reproduced in Fig. 3 where the dimensionless Morton (M), Eötvös (Eo) and Reynolds (Re) are given by:

\[ M = \frac{g \mu_1 \Delta \rho}{\rho_1 \sigma}, \quad (19) \]

\[ Eo = \frac{g \Delta \rho d^2}{\sigma}, \quad (20) \]

\[ \text{Re}_b = \frac{\rho v_d d}{\mu}, \quad (21) \]

**Figure 3:** Bubble diagram of Grace (1973) for the shape and terminal rise velocity of gas bubbles in quiescent viscous liquids.

Figure 4: Rise of a single gas bubble (initial diameter 0.02 m, Eo=38.8, M=9.71·10^{-4}) through a suspension of N_p solid particles (See Table 3 for additional data). For
each series of four frames (from left to right) $t=0.05, 0.10, 0.15$ and $0.20$ s. From top to bottom $N_p$ increases from 0 to 100,000 in these series of frames.

<table>
<thead>
<tr>
<th>Computational grid</th>
<th>40x40x100 (-)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid size</td>
<td>0.001 m</td>
</tr>
<tr>
<td>Number of particles</td>
<td>0 (a) 20,000 (b) 40,000 (c) 60,000 (d) 80,000 (e) 100,000 (f)</td>
</tr>
<tr>
<td>Time step</td>
<td>0.0001 s</td>
</tr>
<tr>
<td>Initial bubble radius</td>
<td>0.01 m</td>
</tr>
<tr>
<td>Initial bubble position</td>
<td>(x_0,y_0,z_0) = (0.02, 0.02, 0.02) m</td>
</tr>
<tr>
<td>Particle density</td>
<td>3000 kg/m$^3$</td>
</tr>
<tr>
<td>Particle diameter</td>
<td>400 μm</td>
</tr>
<tr>
<td>Collision parameters</td>
<td>$c_s$, $c_l$, $μ_t$</td>
</tr>
<tr>
<td>Liquid density</td>
<td>1000 kg/m$^3$</td>
</tr>
<tr>
<td>Liquid viscosity</td>
<td>0.1 kg/(m.s)</td>
</tr>
<tr>
<td>Gas density</td>
<td>10 kg/m$^3$</td>
</tr>
<tr>
<td>Gas viscosity</td>
<td>0.001 kg/(m.s)</td>
</tr>
<tr>
<td>Surface tension</td>
<td>0.1 N/m</td>
</tr>
</tbody>
</table>

**Table 3:** Data used for the simulations shown in Fig. 4.

where the effective diameter $d_e$ is defined as the diameter of a spherical bubble with the same volume as the bubble under consideration. In our simulations a fixed density and viscosity ratio of one hundred was used (viscosity and density of the continuous liquid phase equal one hundred times the viscosity and density of the dispersed gas phase).

First the effect of the presence of solid particles on the rise velocity of isolated gas bubbles is assessed. A typical result is presented in Fig. 4, showing the rise of a single gas bubble through a suspension of $N_p$ solid particles. The particles are initially at rest and are randomly distributed over the bottom section of the domain, excluding the volume occupied by the gas bubble. Initially the bubble is spherical and the suspension is quiescent. For all simulations reported in this paper free-slip boundary conditions were applied at the confining walls. The data used for these simulations are given in Table 1. For this particular case the Eötvös and Morton number, based on the physical properties of the pure (i.e. without solid particles) liquid, are respectively equal to 38.8 and 9.71.10$^{-4}$, which according to the Grace diagram should correspond to a terminal bubble Reynolds number $Re_b$ of 60. For the pure liquid (i.e. without particles) the computed $Re_b$ equals 46 for all cases and is lower than the value according to the Grace diagram. This discrepancy can be attributed to the relatively small lateral size of the container. For the investigated range of solids volume fraction the retarding effect of the particles on the rise velocity of the bubble is quite small, although small changes in the bubble shape can be discerned. From the series of snapshots given in Fig. 4, the particle drift caused by the rising gas bubble can be clearly seen. While the bubble ascends through the suspension the particles are pushed aside and accumulate at the bubble base.

In Fig. 5 the dimensionless volume of the drift zone $X_d$ is plotted as a function of time. Total number of particles: 100,000 (case f given in Table 3).

![Figure 5: Dimensionless volume $X_d$ of the drift zone behind the rising bubble as a function of time. Total number of particles: 100,000 (case f given in Table 3).](image)

$X_d = \frac{N_d V_b}{N_p V_b}$  \hspace{1cm} (22)

where $V_b$ is the initial volume occupied by the suspension. From this figure it can be seen that this dimensionless volume increases steadily and reaches quite high values (corresponding to twice the bubble volume) indicating appreciable transport of the particles due to the passage of the bubble. In Fig. 6 the dimensionless volume of the drift zone $X_d$ is plotted as a function of the solids volume fraction $ε_s$ (cases b to f in Table 1) 0.30 s after bubble release. Apparently the bubble becomes less effective with respect to the vertical particle transport with increasing solids volume fraction.

![Figure 6: Dimensionless volume $X_d$ of the drift zone as a](image)
function of the solids volume fraction $\varepsilon_s$ at $t=0.30$ s after bubble release.

**Figure 7:** Top: snapshots at different times ($t=0.05$, 0.10 and 0.15 s) of the co-axial coalescence of two initially spherical bubbles of 0.010 m diameter released from positions $(0.020 \text{ m}, 0.020 \text{ m}, 0.010 \text{ m})$ and $(0.020 \text{ m}, 0.020 \text{ m}, 0.025 \text{ m})$ in an initially quiescent liquid in a square column of $0.04 \text{ m} \times 0.04 \text{ m} \times 0.08 \text{ m}$, using a $80\times80\times160$ grid and a time step of $5.10^{-5}$ s. $Eo=16$ and $M=2.10^{-4}$. Bottom: identical calculation with only the leading bubble.

**Multiple gas bubbles**

The first example involves the co-axial coalescence of two gas bubbles in an initially quiescent suspension where the bubbles are initially spherical with their centres separated by three bubble radii. In Fig. 7 the computed evolution of the bubble shapes is shown before and after the coalescence process. The Morton and Eötvös number for this case are respectively equal to $2.10^{-4}$ and 16 (based on the individual bubbles) which, according to the Grace diagram, would correspond with a terminal Reynolds number of 50 (case indicated with an asterisk in Fig. 3) which agrees reasonably well with the experimental value of 43 and the computed value of 40. Apparently the leading bubble experiences very little effect of the trailing bubble. However, the behaviour of the trailing bubble is completely different as can be seen clearly from Fig. 7. The trailing bubble catches up with the leading bubble and experiences considerable shape deformation when it enters the wake region of the leading bubble.

The second example involves the oblique coalescence of two gas bubbles in an initially quiescent suspension where the bubbles are initially spherical with their centres separated again by three bubble radii in the vertical direction. Compared to the first case the position of the lower bubble was shifted to the right ($x$-direction) over 1.6 bubble radii. In Fig. 8 the computed evolution of the bubble shapes is shown before and after the coalescence process. The Morton and Eötvös number are the same as for the first case. Similar to the previous case the leading bubble behaves as an isolated bubble although its major (vertical) axis attains a small angle with the (vertical) $z$-axis when the trailing bubble enters its wake region. Again the trailing bubble catches up with the leading bubble and experiences considerable shape deformation when it enters the wake region of the leading bubble.

**Figure 8:** Top: snapshots at different times ($t=0.05$, 0.10 and 0.15 s) of the oblique coalescence of two initially spherical bubbles of 0.010 m diameter released from positions $(0.020 \text{ m}, 0.020 \text{ m}, 0.025 \text{ m})$ and $(0.028 \text{ m}, 0.020 \text{ m}, 0.010 \text{ m})$ in an initially quiescent liquid in a square column of $0.04 \text{ m} \times 0.04 \text{ m} \times 0.08 \text{ m}$, using a $80\times80\times160$ grid and a time step of $5.10^{-5}$ s. Bottom: identical calculation with only the leading bubble.

**Figure 9:** Dimensionless volume $X_d$ of the drift zone behind the rising bubble as a function of time for co-axial bubble coalescence (squares), oblique bubble coalescence
(circles) and the case with the leading bubble only (triangles).

In Fig. 9 the results of these two calculations are compared in terms of $X_d$ computed as a function of time. For the purpose of reference the results of the calculation with the leading bubble only are included in this figure. Surprisingly the case with co-axial bubble coalescence leads to the most pronounced vertical particle transport in the bubble wakes.

CONCLUSIONS

In this paper a hybrid model has been presented enabling the direct numerical simulation of gas-liquid-solid flows using a combined Volume of Fluid (VOF) and Discrete Particle (DP) approach applied for respectively dispersed gas bubbles and solid particles present in the continuous liquid phase. The hard sphere DP model, originally developed by Hoomans et al. (1996) for dense gas-solid systems, has been extended to account for all additional forces acting on particles suspended in a viscous liquid and has been combined with the VOF model presented recently by van Sint Annaland et al. (2005) for complex free surface flows. Our model generates very detailed information on bubble induced particle mixing and can in principle be used to study and quantify the effect of all important physical properties of the gas-liquid-solid system and other variables (such as bubble size and shape) as well. The model has been successfully applied to compute the particle drift behind rising gas bubbles and revealed that in case multiple bubbles rise, the bubbles compete with each other for the particles drifting behind them. Clearly computational resources constitute a limiting factor at present. Our model is quite general and it can in principle also be used to study the behaviour of small bubbles (replacing the particles) in the presence of large bubbles provided that appropriate closure equations are used for the (effective) forces acting on the small (unresolved) bubbles.

REFERENCES


