Two-phase simulations of an off-nominally operating dissolved-air flotation tank

V. Emmanouil, E.P. Skaperdas,
T.D. Karapantsios and K.A. Matis*

Division of Chemical Technology,
School of Chemistry, Aristotle University of Thessaloniki,
University Box 116, 54124, Thessaloniki, Greece
E-mail: vemmanou@chem.auth.gr E-mail: eskape@winweb.gr
E-mail: karapant@chem.auth.gr E-mail: kamatis@chem.auth.gr
*Corresponding author

Abstract: The flow patterns of the contaminated water and the air bubbles inside a large-scale Dissolved Air Flotation (DAF) tank were examined using CFD modelling. The DAF tank was operating off-nominally in the sense that it was supplied with bubbles larger than originally designed for. Attention was drawn to the location for air injection and the effect of having more than one-size bubbles in the tank, regarding the good contact between phases and the subsequent bubble recovery at the tank free surface. The role of the inclination of the internal baffle of the tank and liquid flow rate was also examined.

Keywords: computational fluid dynamics; CFD; dissolved air flotation; DAF; large bubbles; modelling; multiphase models.

Reference to this paper should be made as follows: Emmanouil, V., Skaperdas, E.P., Karapantsios, T.D. and Matis, K.A. (2007) 'Two-phase simulations of an off-nominally operating dissolved-air flotation tank', *Int. J. Environment and Pollution*, Vol. 30, No. 2, pp.213–230.

Biographical notes: Valesia Emmanouil obtained a Diploma in Chemical Engineering from the Aristotle University of Thessaloniki, Greece, in 2000. She also holds an MSc from the same university (2004). Her research interests are in the area of computational fluid dynamics.

Evangelos P. Skaperdas obtained an Honours ME in Aeronautical Engineering from Imperial College London, in 1994. He was awarded an MSc from Cranfield University in the UK, in 1995. His main research interest is in the area of computational fluid dynamics, with ten years experience in various engineering applications.

Thodoris D. Karapantsios obtained an Honors Diploma in Chemical Engineering in 1986 and a PhD Degree in 1993 from the Aristotle University of Thessaloniki. He also holds a MS Degree from the University of Rochester, USA in 1989. He is currently an Assistant Professor at the Department of Chemistry of the Aristotle University of Thessaloniki. His main research interests are in the field of multiphase dynamics. He has published more than 40 papers in refereed journals, participated in more than 30 scientific meetings and has received one European patent. He has coordinated or participated in more than 20 national and international research programs.

Kostas A. Matis obtained his first degree in Chemistry (1973) from the same Department in Aristotle University that he has been serving since 1980. He was awarded his MSc by research (1975) and his PhD (1977) from the Chemical Engineering Department of the University of Newcastle-upon-Tyne, UK. He became a Professor of Chemical Technology, in 1995. He has published more than 120 papers in refereed journals (which have attracted around 600 citations) and participated in more than 50 scientific meetings. He is the editor of three books, mainly on flotation and has received two patents. His main specialisation is in separation science and technology, and also in wastewater treatment, environmental biotechnology, inorganic materials and mineral processing.

1 Introduction

Flotation technology is applied in water and wastewater treatment. Solid particulate contaminated water enters the flotation tank where small diameter air bubbles are injected and mixed with it. Flow mixing enhances bubble dispersion and keeps the solid particles in suspension although the suspension may not be uniform, with the larger heavier particles tending to remain in the lower parts of the tank. Rising bubbles collide with suspended particles and adhere to those that have the appropriate surface characteristics. Each bubble can have many encounters with particles and can carry many of them during its rise. As long as there is net buoyancy force acting on the bubble/particles aggregates, they can separate from the main flow and collect at the free surface of the tank where they are subsequently removed. As the concentration of bubbles, particles and bubble/particles aggregates in the flotation tank is low, water flow is dominant and determines the general flow pattern in the tank. Yet, the movement of bubbles, particles and aggregates creates drag on the water flow and vice versa, the extent of which influences flotation performance (King, 2001; Nguyen and Schulze, 2004).

The flotation tank can be thought of as consisting of two basic regions: a 'noisy' reaction zone and a 'quiet' flotation zone. In the reaction zone, released bubbles are brought vividly in contact with the incoming contaminated water. Particles and bubbles attach in this region. In the flotation zone, the bubble/particles aggregates are allowed to rise, thus separating the solids from the main water stream.

Following the evolution of CFD codes in the last decade, several investigators have employed numerical simulations to examine various aspects of flotation processes. Owing to the exceeding computational burden, most of them refer to two-phase (gas–liquid) systems and only a few attempts have been made with three coexisting phases (gas–liquid–solid). Fawcett (1997) studied the two-phase hydraulics of a DAF tank (average bubble size 70 μm) in a 2D frame of reference using the CFX 4 code. Among the examined parameters (tank dimensions, internal baffle arrangements, water flow rate and air flow rate), he found that the injected air-to-water flow momentum ratio was the most critical design parameter dictating the effective mixing of air with the main liquid stream.

Ta and co-workers made decisive steps in CFD application to DAF tanks: In particular, Hague et al. (2000) compared simulations using a k-ε turbulence model against laminar flow simulations and, furthermore, against laser Doppler velocimetry data for single and two-phase flow (average bubble size 50 μm) using FLUENT 4.5 code.

Strangely, at the quiescent flotation zone of the tank, the turbulent model gave closer agreement with the measured horizontal velocities than the laminar model. This peculiarity should be rather attributed to the small size of the employed lab scale tank, which did not allow vortices from the reaction zone to sufficiently decay before entering the flotation zone. In another study, Ta (2000) recognised that the water flow would dominate the motion of air bubbles inside the tank when bubbles are small (20–100 μm) and the air volume fraction is below 10%. In a more advanced effort, Ta et al. (2000) simulated a large scale DAF tank (average bubble size 50 μm) incorporating three phases and a 3D structure grid. Air/water flow was described by a *Eulerian–Eulerian model* whereas particles were tracked using a disperse Lagrangian model. No bubble/particle collision was considered. CFD results were compared with visual information from an underwater camera and an acoustic Doppler velocimetry technique. Overall, the agreement between measurements and predictions was only qualitative with CFD being unable to predict unsteady flow features such as lateral dispersion of bubble clouds.

Working with a dispersed-air flotation tank of special design, the so-called Bubble Accelerated Flotation (BAF) tank, Desam et al. (2000) undertook a detailed CFD study where the model was validated against measurements from a dye tracing and a photographic technique. The Fluent 5.0 code was used to simulate the flotation zone in three dimensions. The effect of bubble diameter as well as the inlet/outlet position and inlet velocity of the air/water mixture in the separation tank, were examined and found to be significant on the separation efficiency. Bubbles 250, 500 and 1000 μm were used (one size at a time) in the calculations. A message from their work is that by proper selection of the system geometrical details and further adjustment of the bubble/liquid flow rates it is possible to achieve appreciable mixing of phases and bubble recovery at the tank surface even under reduced shear conditions. This is the case of a DAF tank where the only shearing action is imposed by the liquid flow as it moves through the reaction zone and mixes with the injected bubbles.

CFD simulations on a Denver-type dispersed-air flotation tank was undertaken by Koh et al. (2000) using the CFX 4.1 code. Solids and liquids were considered as one phase and bubbles as another. The k-ε turbulence model was used to describe the dynamics inside the tank in three dimensions. It was the first time that flotation kinetic concepts such as collision probability and collision efficiency were used in CFD studies. Recently, Koh and Schwarz (2003) included in their simulations a turbulent collision model to estimate *locally* the rate of bubble-particle encounters, from local values of the turbulent velocity and the size and concentrations of bubbles and particles.

From the above it is apparent that the application of CFD codes to simulate large-scale flotation processes offers distinct advantages in investigating the relative significance of various design and operating parameters. At present, the computer load when many phases are solved simultaneously is large and even simple 2D problems take several hours to converge. With this in mind, this paper uses the multiphase models incorporated in the Fluent 6.1 code to describe the hydrodynamics inside a large-scale DAF tank that is ill-functioning because it is supplied with bubbles larger than 50–100 μm (which is the bubble size range for nominal performance). In real situations, such irregularity may occur owing to a defective air injection pressure reducing needle valve, partially blocked air injection nozzles or even a viscosity increase of the continuous phase (O'Neill et al., 1997; Matis, 1981). The aforementioned causes can slow down the de-pressurisation process of the injected gas-saturated stream which, in turn, may slow down bubble nucleation and growth and make newly born bubbles coalesce, eventually leading to larger bubbles in the tank. The relocation of the air injection nozzles in the tank and the coexistence of bubbles of different sizes in the reaction zone are examined as a means to enhance the performance of such a large-bubble laden DAF tank.

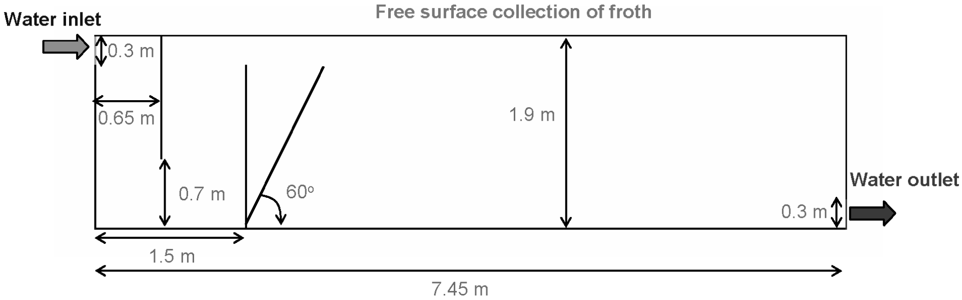
2 Numerical model

Fluid flow in the flotation tank is considered as a two-phase (gas–liquid) flow in which bubbles move inside a continuous aqueous phase. As bubbles travel through the continuous phase, they gain or loose momentum through surface drag and gravity. These dispersed bubbles are considered as particles with air properties (Lagrangian approach). The model does not describe the solids or the bubble/particles aggregates. As the physics of bubble/particles attachment is not well understood it would be misleading at present to include equations for this. Moreover, the run times for the simulations would become impractical. All this means that the performance of the flotation tank has to be inferred solely from the water velocities and air concentrations.

To model the two-phase flow in the present flotation tank, the code FLUENT 6.1 is used. The selection of the model to simulate a two phase-dispersed flow depends on whether the dispersed phase is dense or dilute. Since the dispersed phase is dilute in the flotation tank a Eulerian–Langrangian model (discrete phase model, *DPM*) is selected for the continuous–dispersed phase, respectively. In *DPM*, the dispersed phase is represented by bubbles that are continually introduced at a given mass flow rate and are carried along with the main flow. The interaction between phases is accounted for by adding a momentum source term in the Eulerian conservation equation of the continuous phase. In order to check the results produced by *DPM*, the same simulations were also performed using two *Eulerian–Eulerian models*, the so-called *Mixture* and *Eulerian models* (FLUENT User Guide, 2003). The *Eulerian model* is the most accurate but it is also a demanding model in terms of computational effort as it solves the continuity and momentum equations for each phase separately. The *Mixture model* is a time-saving compromise to the *Eulerian model* that solves the continuity and momentum equations, assuming that all phases behave as a mixture, and then uses the concept of drift and slip velocities to describe the dispersed phases.

The modelling procedure involves two steps. The first step involves building the grid using GAMBIT pre-processor and imposing the boundary conditions on the model. All the model and solution parameters are fixed to obtain a solution using FLUENT solver in the second step. The geometrical details of the flotation tank are shown in Figure 1 as generated in GAMBIT. The main dimensions of the tank are 7.45 metres long and 1.9 metres high. These were the tank dimensions used by Fawcett (1997) and are adopted here for the sake of comparing our results with his. Besides, these dimensions are typical of industrial DAF tanks (O’Neill et al., 1997).

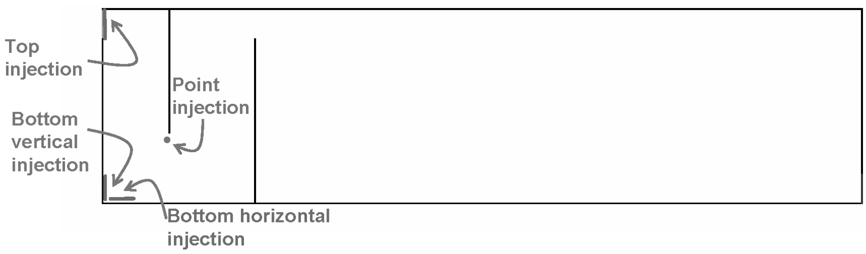
Figure 1 Flotation tank geometry



There are two vertical baffles near the entry of the tank to control the fluid flow in the tank. The one on the left is fixed vertically whereas the one on the right can be set either vertical or sloped at certain angles. Two baffle angles are tested in this work, 60° and 75° (from the horizontal) but results only with the former are presented here because of space limitations. Owing to the baffles' arrangement, the liquid which enters the tank is forced to travel through a downflow section first and an upflow section afterwards. This is where the incoming contaminated water mixes with bubbles. The selection of parameters is made such that it would allow sufficient turbulence to mix phases and provide enough bubble/particle collisions but not too much to avoid bubble coalescence and bubble/particle breaking-up. Most simulations are performed with an inlet liquid velocity of 0.26 m/s (0.09 kg/s) but runs with 0.52 m/s (0.18 kg/s) are also made.

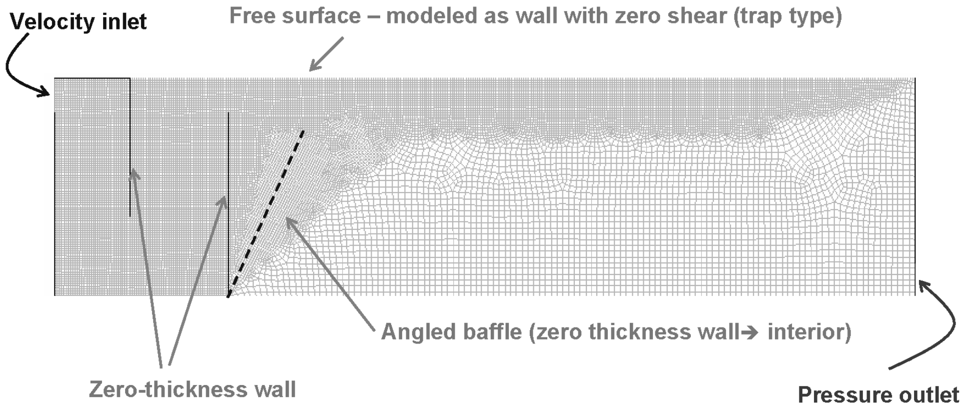
Four different locations for air injection are examined, as shown in Figure 2. The air in a DAF tank is usually injected in the form of a gas-supersaturated recycle stream through a series of nozzles/needle valves at the bottom of the vertical baffle (Fawcett, 1997). Alternatively, in some designs the recycle stream may be injected at the bottom of the reaction zone or into the pipe connecting the flocculation with the flotation tank; in the later case air enters the tank together with the inlet main flow (O'Neill et al., 1997; Ta et al., 2000). Figure 2 includes all these alternatives. Air is introduced as individual bubbles of certain size. Bubble sizes of 200, 500 and 1000 μm are considered entering alone or altogether in the tank. Apart from the *point injection* case, bubbles are introduced from 15 distinct nearby injection points. In every run, 30 bubbles of the same size are fed from each injection point, which for all the three bubble sizes brings the total number of injected bubbles (per run) to 1350. The gas flow rate is set at 0.1 kg/m^3s (4.9×10^{-3} kg/s) a value warranting that the local air volume fraction is always below ~10% inside the tank, as recommended for the application of *DPM* (FLUENT User Guide, 2003).

Figure 2 Bubble injection points



The boundary condition for the tank inlet is the prescribed inlet water velocity with 10% turbulence intensity. The boundary condition for the outlet is the pressure at the exit of the tank with 10% backflow turbulence intensity and reference pressure the hydrostatic value (Figure 3). In the case of the Eulerian–Lagrangian model, the free surface of the tank is modelled as a wall with zero shear (trap type) whereas in the *Eulerian–Eulerian models*, the free surface is considered as simply in contact with the air phase. The remaining walls are modelled as fixed walls with a zero near surface velocity.

Figure 3 Grid of flotation tank



The standard *k*-*ε* turbulence model is used to describe the turbulence inside the tank. This model is shown to be particularly effective in describing bubble dynamics in situations with instationary turbulent character (Sokolichin and Eigenberger, 1999; Borchers et al., 1999). In addition, it converges more readily than the other turbulence models. The implicit segregated model is employed to solve the governing equations while the pressure-velocity coupling is discretised by the SIMPLE algorithm.

In order to allow comparisons in reasonable time, simulations are performed in two dimensions, which still allow much to be learnt. The *base case* below refers to an inlet liquid velocity of 0.26 m/s and a gas flow rate of 0.1 kg/m^3s with the internal baffle vertically positioned. Results are presented as 2D contours of water velocity and air volume fraction inside the tank. When necessary, bubble trajectories are also shown.

3 Results

3.1 Single phase vs. two-phase flow

Figure 4 is a plot of the single phase (water only) flow pattern for the *base case*. Figure 5 is the same plot but with the second phase, air, included (200, 500 and 1000 μm bubbles, introduced horizontally at the bottom of the reaction zone). Comparing the two figures, it is striking how different the flow patterns are in the reaction zone. The injection of bubbles creates a broader spread of velocities (uniform flow) within the reaction zone as a result of large-scale turbulence and mixing which is not present in the single-phase flow. This is chiefly because of the strong drag and significant buoyant force imposed by large bubbles that yield higher local velocities and prevent the water tendency from following the line of the second (internal) baffle. Also, the presence of bubbles keeps the velocity near the bottom of the tank high so that suspended particles can not deposit easily. These observations agree qualitatively with the findings of Fawcett (1997) in a similar DAF tank operating with much smaller bubbles (50 μm). Yet, our larger bubbles (in fact, bubble/particle aggregates) can reach more readily the surface of the tank and so it is less probable that some of them will escape through the water outlet to the filters after the tank.

Figure 4 Velocity magnitude (m/s) contours in single flow (water)

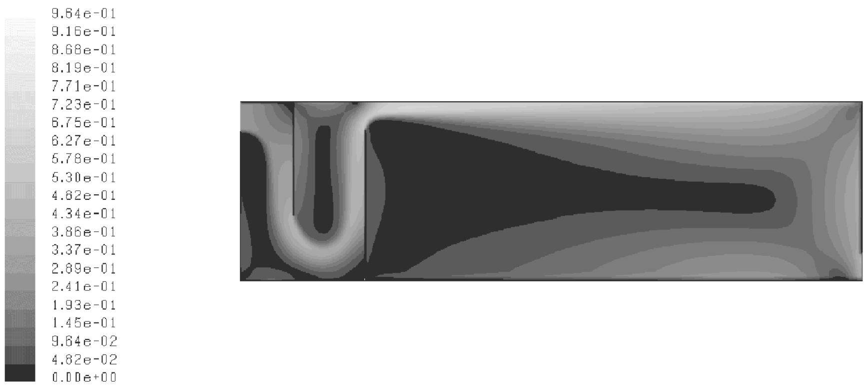
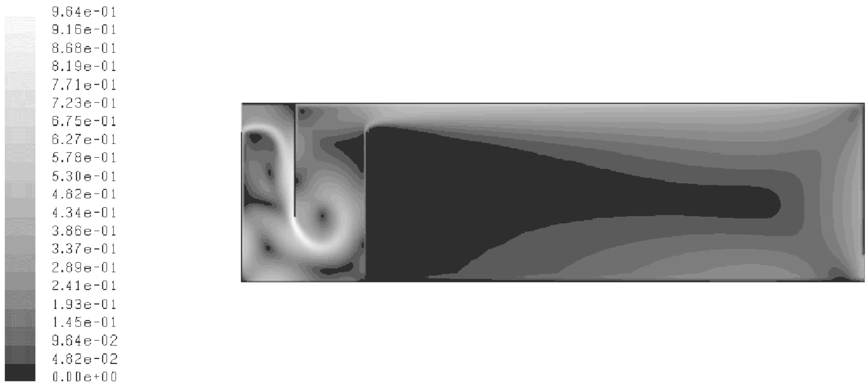


Figure 5 Velocity magnitude (m/s) contours of water in two-phase flow (water-bubble)



3.2 Comparison of multiphase models

In the absence of field measurements, one way to indirectly judge on the adequacy of multiphase models is to compare results among different modelling schemes. The most stringent comparison is between the *DPM* and the rigorous *Eulerian model*. It must be noted that in our case the Eulerian and the *Mixture models* produces virtually the same results. This verifies the multiphase model selection criteria suggested by FLUENT, which clearly indicates that the *Mixture model* is as suitable as the *Eulerian model* for our applications. However, for the sake of increasing the rigorousness in predictions, most presented results are based on the *Eulerian model*.

The comparison between *DPM* and *Eulerian model* is presented in Figures 6–9 with respect to the distribution of water velocity and air volume fraction inside the tank. With the *Eulerian model* it is imperative to have the inlet and outlet of the flow not directly imposed to the tank but instead use a phantom entrance and exit pipeline to overcome the abrupt disturbance of the velocity profile at these points. Figures 6 and 7 refer to the *base case* and bubbles 1000 microns in diameter. Figures 8 and 9 refer to an inlet water velocity of 0.52 m/s and 200 microns bubbles. The above represent two important limiting cases for the operation of the tank.

Figure 6 Velocity magnitude of water (m/s) with 0.26 m/s water inlet velocity and bubbles 1000 μm: (a) DPM and (b) Eulerian model

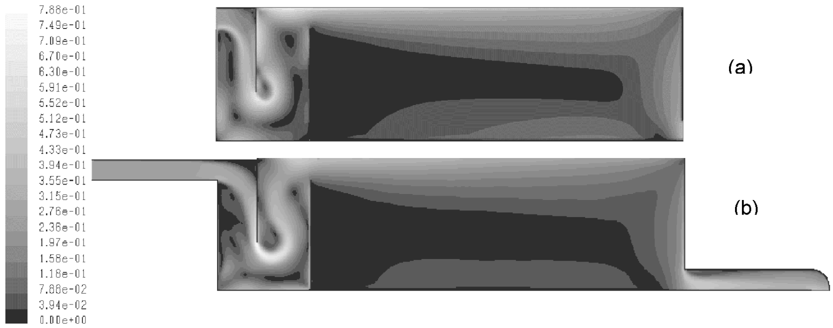


Figure 7 Volume fraction of air with 0.26 m/s water inlet velocity and bubbles 1000 μm: (a) DPM and (b) Eulerian model

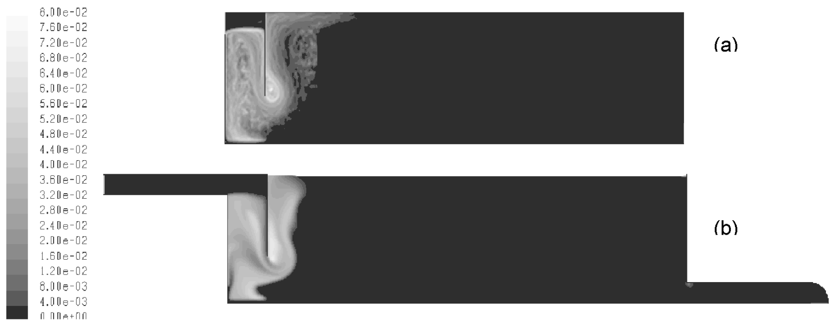


Figure 8 Velocity magnitude of water (m/s) with 0.52 m/s water inlet velocity and bubbles 200 μm: (a) DPM and (b) Eulerian model

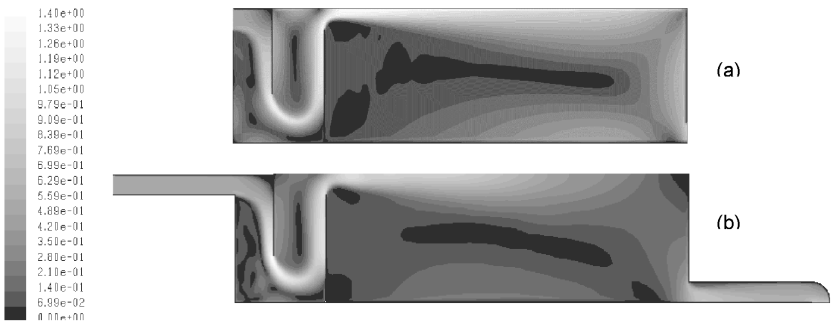
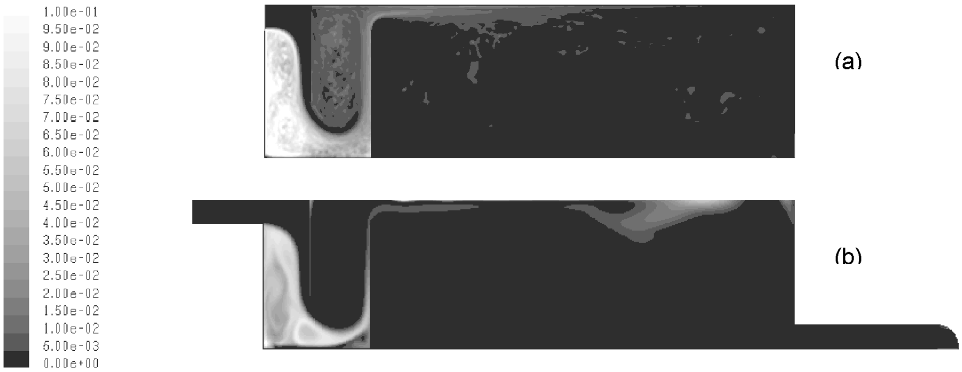


Figure 9 Volume fraction of air with 0.52 m/s water inlet velocity and bubbles 200 μm: (a) DPM and (b) Eulerian model



As regards the velocity magnitude in Figure 6, some difference is observed at the free surface of the flotation zone but this is rather because of the different boundary conditions employed for the two models. Inside the reaction zone, the *Eulerian model* predicts an upflow towards the free surface of the tank not so confined near the first vertical baffle. However, the general velocity distribution patterns inside the tank are quite similar. Regarding the air volume fraction in Figure 7, the *Eulerian model* gives a distinctly narrower spread of bubbles in the upflow region of the reaction zone. Closer inspection of the *DPM* results reveals that the volume fractions in the areas of high spread of (trapped) bubbles does not exceed ~1–2%. Judging from the shape of the air distribution and also the bubble trajectories (not shown) in these areas, one can argue that the additional spread of bubbles predicted by *DPM* is rather a computational artifact owing to the inability of the steady-particle Langrangian *DPM* to correctly describe bubbles that are trapped in iterative flow loops (FLUENT User Guide, 2003). Apart from that, *DPM* appears to give comparable results with the *Eulerian model* in the regions of high significance, i.e., where high velocities and air volume fractions are encountered.

A satisfactory agreement between the two models regarding the overall velocity and air distributions is noticed also in Figures 8 and 9. The employed high liquid velocity clearly reduces the effect of small bubbles in the liquid flow, Figure 8. Yet, a discrepancy is noticed in Figure 9 regarding the magnitude of the local air volume fractions. It is apparent that *DPM* predicts unrealistically high fractions of 200 microns bubbles owing to the fact that many of them are trapped in iterative loops at the leftmost region of the reaction zone where the liquid velocity has only very low values to compete with buoyancy. All in all, *DPM* seems to give reliable results for most of the cases in this study with reservations only when small bubbles are trapped indefinitely in flow loops. Thus, henceforth *DPM* will be used for assessing various designs and operating parameters of the tank.

3.3 Effect of bubble injection location

Air injection is examined at the four different locations shown in Figure 2. The optimum location is selected based on the extent of bubbles dispersion inside the reaction zone of the tank combined with a substantial recovery of bubbles at the free surface at the flotation zone of the tank. Figures 10–13 display the trajectories of all the bubbles employed in this work, 1000, 500 and 200 microns, namely. In these plots, the bubbles are all simultaneously fed in the tank. It is apparent, that the broadest bubble dispersion in the reaction zone is obtained when the bubbles are introduced at the bottom (vertical or horizontal) of the tank. In addition, for the latter two locations, the bubble dispersion seems adequate in terms of uniform spreading and mixing with the liquid flow. This observation holds regardless possible errors owing to *DPM* inadequacy to describe bubbles in flow loops. For the same DAF tank operating with bubbles less than 100 microns, Fawcett (1997) found that sufficient turbulent mixing of bubbles with the liquid stream was achieved even with air injected from a single point at the lower end of the rigid vertical baffle. It must be stressed though that employing a point injection in a two-dimensional grid is not strictly correct because bubbles are forced to pass through a narrow region producing unrealistically high gas volume fractions and thus violating the requirement of *DPM* to work at fractions below ~10% (FLUENT User Guide, 2003).

Figure 10 Bubble trajectories: top injection

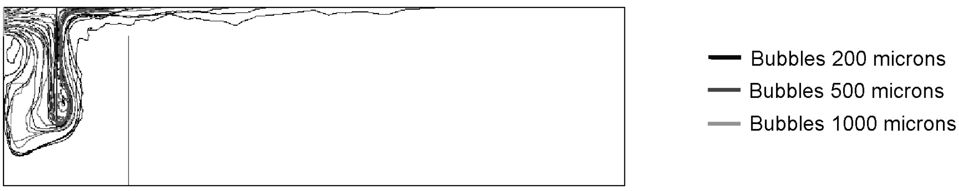


Figure 11 Bubble trajectories: point injection

Figure 12 Bubble trajectories: bottom vertical injection

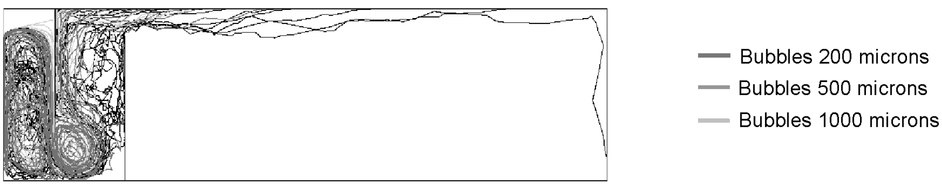
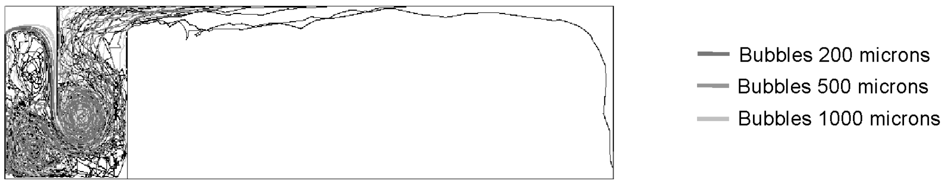


Figure 13 Bubble trajectories: bottom horizontal injection



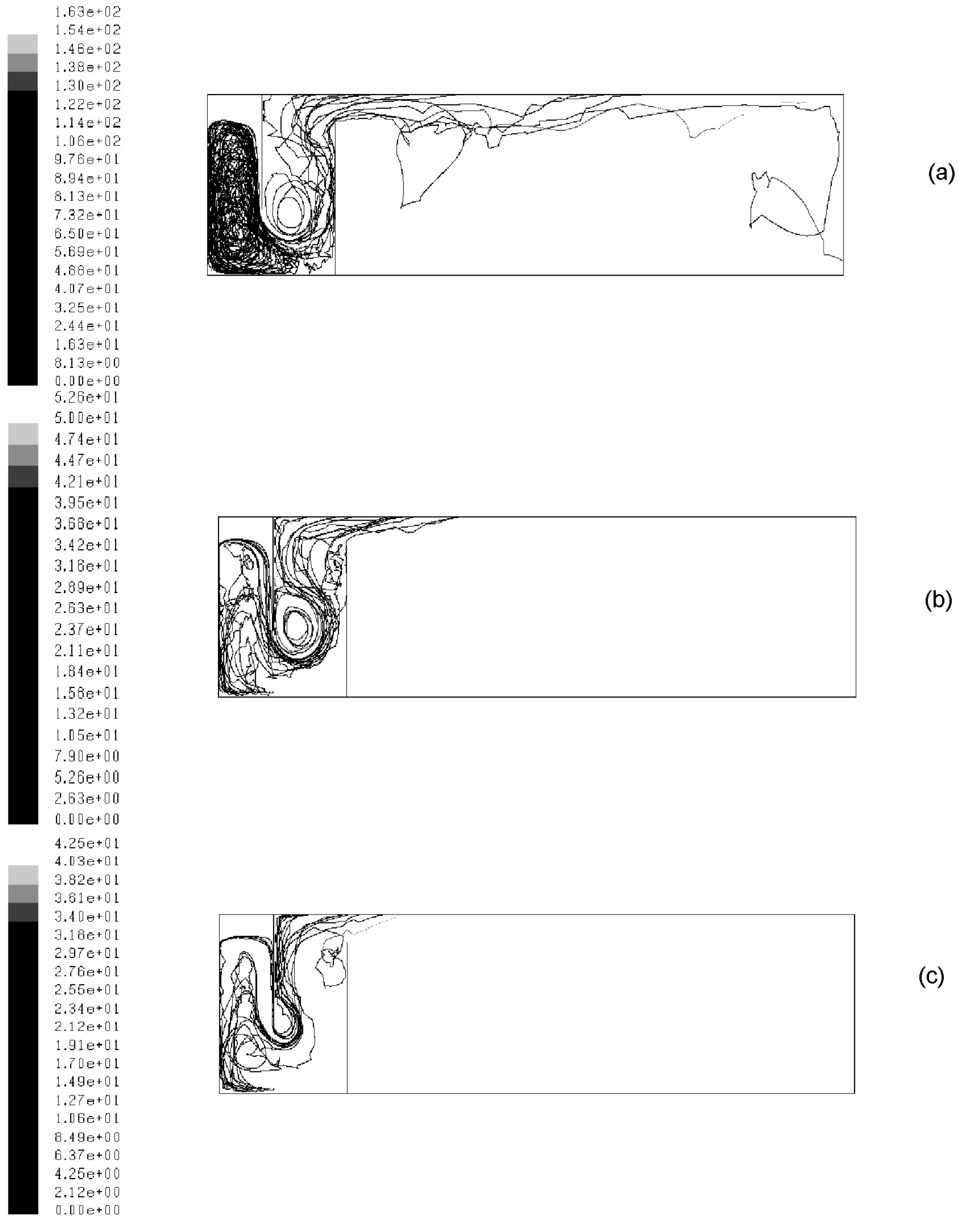
The large bubbles of this work rose readily to the surface of the tank because of the high buoyancy forces and their resistance to the momentum of the water flow. This feature, in general, leads to poorer mixing between phases but a better bubble recovery at the free surface of the tank. The case where bubbles are fed at the bottom horizontal location presents better recovery characteristics for all the employed bubble sizes. That's why for the rest of this study, bubbles are introduced from that location only. Apparently, for the irregular situation where large bubbles are supplied to the DAF tank, bubble/liquid mixing and bubble recovery can still be adequately accomplished if air injection takes place at the bottom of the reaction zone. This location for air injection is not unusual even in nominally operating DAF tanks (O'Neill et al., 1997).

3.4 Effect of bubble size

In cases that the air injection system of a DAF tank is not working properly it is very probable that more than one bubble size will be produced. To investigate this effect, three sets of bubbles with diameters of 200, 500 and 1000 microns are examined to flow separately or altogether inside the tank. For the case of smaller bubbles, Figure 14(a) and 14(b), the buoyancy forces are not enough to prevail over the water momentum and the bubbles are easily carried along by the water flow. On the contrary, the bigger bubbles rise pretty rapidly to the tank surface because of the high buoyancy forces which overcome the dissipative turbulent eddies in the water, Figure 14(c).

In Figure 14, the lighter the colour of the trajectory, the longer the lifetime of the bubble. It is evident that the smaller bubbles (200 μm) travel for longer times in the bulk of the tank before surfacing, thus, offering a higher probability of collision with suspended particles (Nguyen and Schulze, 2004). On the other hand, these bubbles are influenced more by the water stream and their relative velocity with respect to the carried-over particles is small. The latter has a negative influence on the rate of collision between bubbles and particles. The overall efficiency of collision (net effect) is the product of the probability times the rate of collision (King, 2001). For the smaller bubbles (200 μm) there is also a chance that some of them may be dragged down to the tank outlet, Figure 14(a). This is highly undesirable as it means that particulate matter attached to these bubbles will escape from the tank.

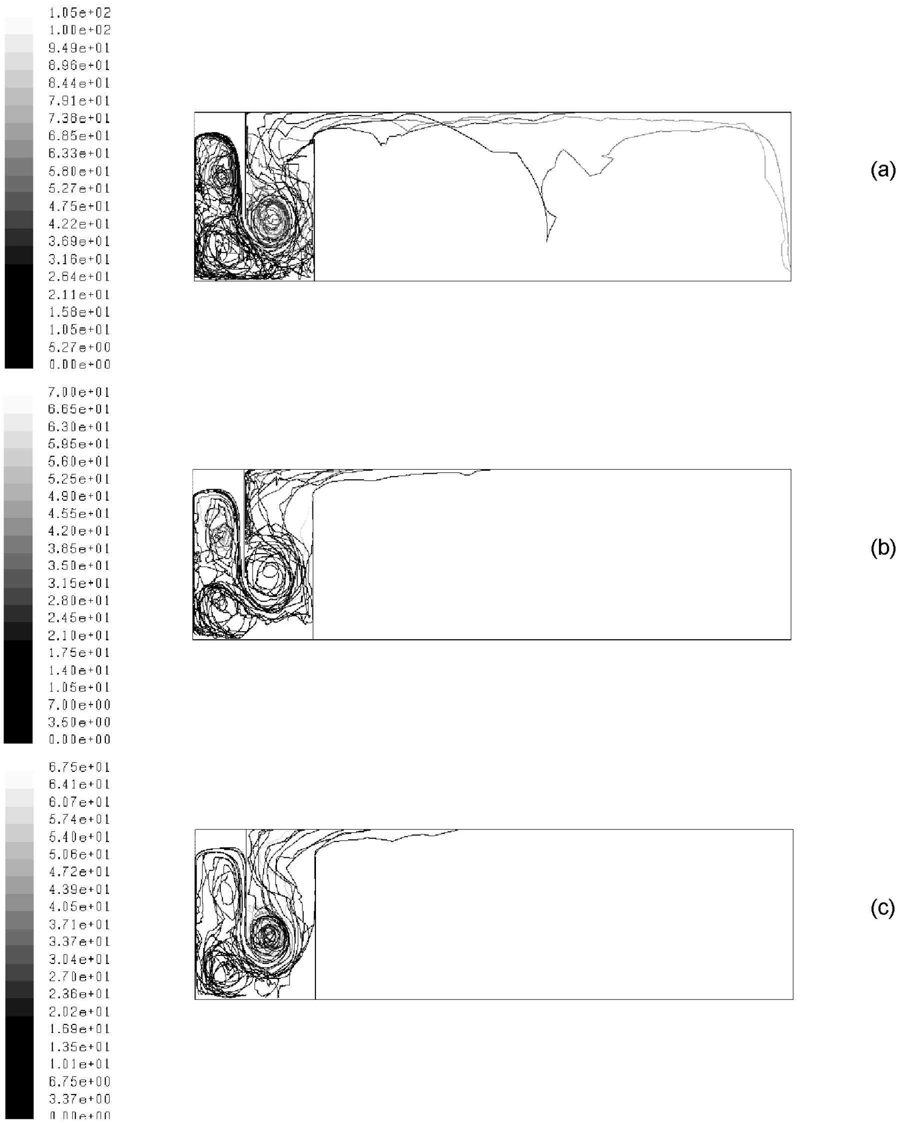
Figure 14 Trajectories of: (a) 200 µm bubbles; (b) 500 µm bubbles and (c) 1000 µm bubbles. Time scale in seconds



The mixing pattern inside the reaction zone is also influenced by the bubble size. The best mixing occurs for 500 µm bubbles, which, for the employed water flow rate, appears to swirl better and disperse more in the available space. Moreover, for this size the bubbles spread quite uniformly over the free surface above the reaction zone of the tank. This is a favourable condition for flotation as floc particles would rather build a uniform froth layer.

Next, the simultaneous presence of the three bubble sizes in the tank is examined, Figure 15. The time scales of the plots show that the 200 μm bubbles are speeded up, the 1000 μm bubbles are slowed down whereas, the 500 μm bubbles remain practically unaffected. Yet, in all cases, bubble trajectories changed appreciably and their dispersion and recovery are better than with single size bubbles. Thus, for an ill-functioning DAF tank (operating with larger than 100 μm bubbles) the unintentional presence of a broad distribution of large bubble sizes appears to improve bubble/liquid mixing and bubble recovery.

Figure 15 Trajectories of: (a) bubbles 200 μm; (b) bubbles 500 μm and (c) bubbles 1000 μm when the three sizes coexist in the tank. Time scale in seconds

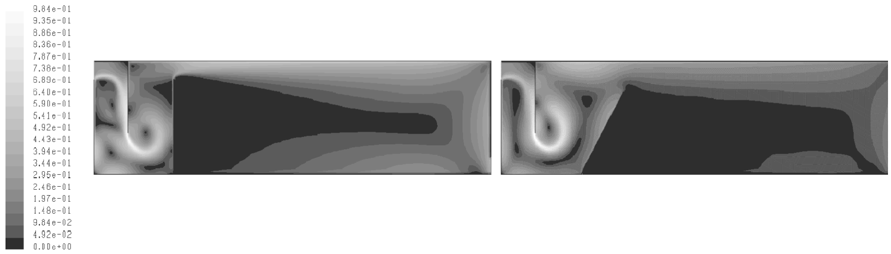


3.5 *Effect of the internal baffle inclination*

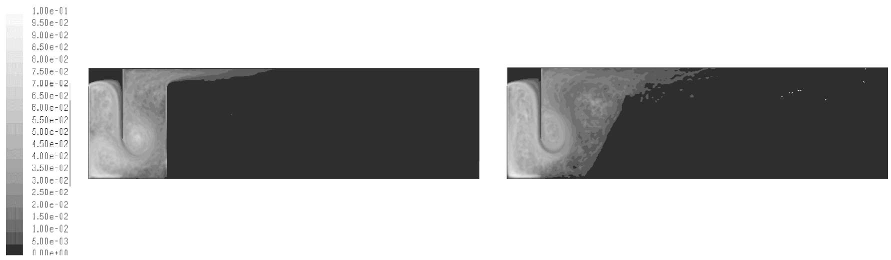
It is mentioned in literature (Lundh et al., 2002; Fawcett, 1997; O'Neill et al., 1997) that one of the most important design parameter of DAF tanks is the position of the internal baffle. An angled internal baffle would reduce the velocities in the reaction zone (by increasing the cross section of the flow path), leading to a longer residence time of bubbles in the reaction zone, thus, a longer time for collisions with particles. Of course, such a feature is desirable only if the intensity of turbulence and blending of the two phases in the reaction zone are already above adequate. Apart from that, O'Neill et al. (1997) argued about non-uniform or recirculating flows induced by baffles angled more than 70° from the horizontal. This, however, refers to bubbles less than 80 μm which have the natural tendency to follow the main water flow: moving up along the sloping baffle and then coming back down towards the inside of the reaction zone. For the larger bubbles of this study where buoyancy and drag forces play a key role, bubble trajectories are different.

Figure 16 presents results of the *base case* for a vertical and an inclined internal baffle at 60° from the horizontal and for the three sizes of bubbles coexisting in the tank. The velocity magnitude, air volume fraction and bubble trajectories of the two scenarios are displayed.

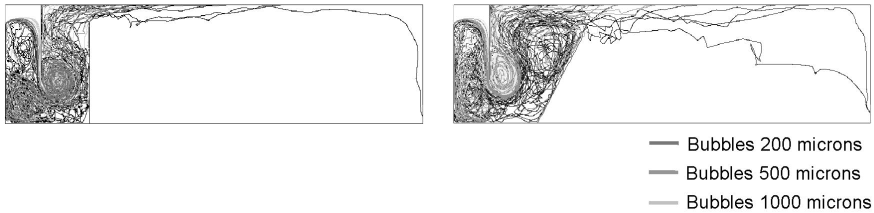
Figure 16 (a) Velocity magnitude (m/s) of water; (b) air volume fraction and (c) bubble trajectories of the vertical and 60° inclined baffle



(a)



(b)



(c)

From Figure 16(a) it is evident that when the baffle is inclined the velocity gradients are smoothed out and so a more uniform flow leaves the reaction zone. Interestingly, the rapid narrow stream clinging off downwards from the fixed vertical baffle is essentially unaffected owing to the appreciable liquid flow rate. The velocities near the bottom of the reaction zone are somewhat lower with the inclined baffle, indicating that large bubble/particles aggregates may deposit at the bottom of the tank. This is highly undesirable for it would require the tank to be drained and cleaned on a regular basis. Lower velocities are also noticed at the free surface of the tank with the inclined baffle, which is because of the wider area for flow separation at the top of the inclined baffle as compared to the area available above the vertical baffle. This is in favour of the inclined baffle arrangement.

The air volume fraction, Figure 16(b), in the upflow section of the reaction zone is less uniformly distributed for the inclined baffle where also a low-gas content recirculation loop is witnessed close to upper part of the baffle. This means that the main flow moves from the reaction zone to the flotation zone through a narrower area (close to the right side of the fixed vertical baffle). This is a disadvantage of the inclined baffle arrangement. The above arguments are further supported by the trajectories shown in Figure 16(c), which seem to be more uniformly distributed for the case of the vertical baffle. On the whole, for the employed input parameters the inclined baffle option offers more drawbacks than advantages. It must be recalled here that *DPM* does not provide accurate solutions for recirculating bubbles but this argument does not invalidate the above general observations.

3.6 *Effect of the inlet velocity*

By adjusting the magnitude of the inlet velocity, one can tune the turbulent mixing and phase distributions in the tank. In order to see these effects, the inlet velocity is doubled from the *base case* value to 0.52 m/s, namely. Figures 17 and 18 show results of velocity magnitude and bubble trajectories, respectively, for this increased inlet velocity and for the three sizes of bubbles flowing simultaneously in the tank. Comparisons can be made between Figures 17 and 5 and Figures 18 and 13.

Figure 17 Velocity magnitude of water (m/s) when the velocity inlet is 0.52 m/s

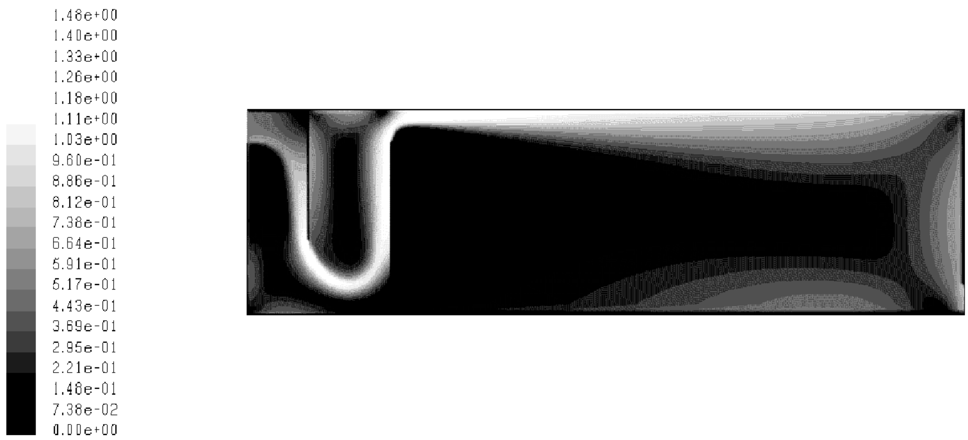
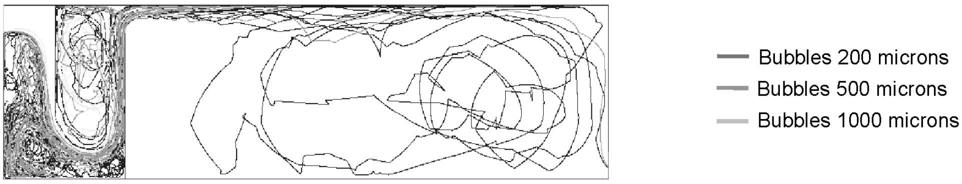


Figure 18 Bubble trajectories when the velocity inlet is 0.52 m/s



With a doubled inlet velocity, a narrow fast moving stream of water clearly develops inside the reaction zone as compared with the base case (Figure 17 vs. Figure 5). This certainly reduces the residence time of liquid and, therefore, the probability of collision between bubbles and particles inside the DAF tank. On the other hand, bubble/water mixing and bubble dispersion are enhanced, Figure 18 vs. Figure 13, but now several bubbles, chiefly of 200 μm and 500 μm, are dragged along by the main flow to the outlet of the tank, such that the tank would be at significant risk of solids carry-over to the filters.

3.7 Limitations of current CFD models

It is important to summarise here the main limitations of the employed CFD models. In order to neglect bubble–bubble interactions, e.g., coalescence and breakage, *DPM* requires the discrete phase (bubbles) volume fractions to be less than 10–12%. Yet, such interactions are less pronounced in our system owing to the absorbed surfactants onto the bubbles surface. Moreover, the employed steady-particle Lagrangian *DPM* does not effectively model flows in which bubbles are suspended indefinitely in the continuum, as occurs for bubbles recirculating within closed loops. On the other hand, the *Mixture model* offers shorter computational times than the *Eulerian model* but it is only the latter that can correctly handle cases of high bubbles loading (even locally) because then there is a two-way coupling between the phases (i.e., the fluid carrier influences the bubbles via drag and turbulence, but the bubbles in turn influence the carrier fluid via reduction in mean momentum and turbulence). To this end, it must be stressed that conclusive statements about the suitability of any CFD code should be based on validation against experimental data that, unfortunately, were not available in our case.

4 Conclusions

The performed CFD simulations were a valuable tool for understanding the off-nominal operation of a DAF tank. The work was done in two dimensions meaning that effects arising from side walls were ignored but still valuable information was gathered at a reasonable computational time. In the absence of experimental data to validate the model, results from three multiphase models incorporated in the Fluent 6.1 code were cross-examined to increase the confidence in predictions. For most of the employed range of input parameters, the predictions of the *DPM* agreed considerably with predictions of the *Mixture model* and the rigorous *Eulerian model*. The main findings of this work were

- The presence of bubbles affected significantly the flow of the liquid stream.
- Four different locations for air injection were examined. The most suitable location to inject large bubbles of 200, 500 and 1000 μm, in the tank was from the horizontal position at the bottom of the tank, which yielded sufficient mixing between phases and satisfactory bubble recovery at the free surface of the flotation zone.
- Bubble trajectories depended on their size: 200, 500 and 1000 μm. The smaller bubbles were affected more by the liquid flow and were easier to get dragged to the tank outlet.
- When all three sizes of bubbles coexisted in the tank, bubbles spread more and occupied more space in the tank compared to what was observed when only one bubble size existed. Furthermore, the simultaneous presence of different bubble sizes had a great impact on the bubble trajectories of every separate single size.
- Putting the internal baffle at an angled position seemed to deteriorate the overall flotation performance for the employed input parameters.
- Increasing the liquid inlet velocity enhanced turbulence mixing of bubbles with liquid but it also reduced the residence time of both phases in the tank while at the same time increased the risk of loosing material (bubble/particles aggregates) through the tank outlet.

References

Borchers, O., Busch, C., Sokolichin, A. and Eigenberger, G. (1999) 'Applicability of the standard k-ε turbulence model to the dynamic simulation of bubble columns. Part II: Comparison of detailed experiments and flow simulations', *Chem. Eng. Sci*., Vol. 54, pp.5927–5935.

Desam, P.R., Datta, A., Moerse, W. and Miller, J.D. (2000) 'A computational fluid dynamics (CFD) model for a bubble separation tank used in wastewater treatment', *Proc. XXI Intl. Mineral Proces*., Congress, Rome, pp.1–12.

Fawcett, N.S.J. (1997) 'The hydraulics of flotation tanks: computational modelling', *Dissolved-Air Flotation, Proc. Intl. Conf. Chartered Inst. Water and Envir. Management*, London, pp.51–71.

FLUENT User's Guide (2003) Version 6.1, Fluent Inc, on CDROM.

Hague, J., Ta, C.T., Biggs, M.J. and Sattary, J.A. (2000) 'Small scale model for CFD validation in DAF application', *Proc. 4th Intl. Conf. Flotation in Water and Waste Water Treatment*, Helsinki, Session 6.

King, R.P. (2001) *Modelling and Simulation of Mineral Processing Systems*, Butterworth-Heinemann, pp.289–350.

Koh, P.T.L. and Schwarz, M.P. (2003) 'CFD modelling of bubble-particle collision rate and efficiencies in flotation cells', *Minerals Eng*., Vol. 16, pp.1055–1059.

Koh, P.T.L., Manickam, M. and Schwarz, M.P. (2000) 'CFD simulation of bubble-particle collisions in mineral flotation cells', *Minerals Eng*., Vol. 13, pp.1455–1463.

Lundh, M., Jönsson, L. and Dahlquist, J. (2002) 'The influence of contact zone configuration on the flow structure in a dissolved air flotation pilot plant', *Water Res*., Vol. 36, pp.1585–1595.

Matis, K.A. (1981) *Flotation and its Application in Industrial Wastes Treatment*, Chimika Chronika – Gen. Ed., Vol. 46, pp.27–33 (in Greek).

Nguyen, A.V. and Schulze, H.J. (2004) *Colloidal Science of Flotation*, Dekker, New York.

O'Neill, S., Yeung, H. and Oddie, G. (1997) 'Physical modelling study of the dissolved air flotation process', *Dissolved-Air Flotation, Proc. Intl. Conf. Chartered Inst. Water and Envir. Management*, London, pp.75–81.

Sokolichin, A. and Eigenberger, G. (1999) 'Applicability of the standard *k*-*ε* turbulence model to the dynamic simulation of bubble columns. Part I: Detailed numerical simulations', *Chem. Eng. Sci*., Vol. 54, pp.2273–2284.

Ta, C.T. (2000) 'Application of Computational Fluid Dynamics to water industry', *Proc. 4th Intl. Conf. Flotation in Water and Waste Water Treatment*, Helsinki, Session 6.

Ta, C.T., Beckley, J. and Eades, A. (2000) 'A multiphase CFD model of DAF process', *Proc. 4th Intl. Conf. Flotation in Water and Waste Water Treatment*, Helsinki, Session 6.