9th IWA Symposium on Systems Analysis and Integrated Assessment
14-1 June 2015, Gold Coast, Australia

A simple CFD approach for the simulation of the flow in dissolved air flotation tanks

Sesma-Pardo, I.[1], Rivas, A.[1] Lizarralde, I.[1,2], Peral, J.[3], Arruabarrena, I.[4], Amores J.[5], Ayesa E.[1,2], Grau, P.[1,2] and Larraona, G.S.[1]

[1] TECNUN University of Navarra. Paseo Manuel Lardizábal 13, 20018 San Sebastián, Spain.
[2] CEIT. Paseo Manuel Lardizábal 15, 20018 San Sebastián, Spain.
[3] LEORPE S.L. Polígono Ibarluze s/n, 20120 Hernani, Spain.
[4] TECEXSA S.A. Polígono Ibarluze s/n, 20120 Hernani, Spain.
[5] EGILE CORPORATION XXI. Poligono Industrial Kurutz Gain 12-13, 20850 Mendaro, Spain.

Keywords: CFD; Dissolved air flotation; Simulation

Summary of key findings

The performance of a Dissolved Air Flotation (DAF) tank was analyzed by Computational Fluid Dynamics (CFD). The three-phase flow (water, bubbles and aggregates) was simulated using a simplified and economical computational approach. The influence of different parameters was studied through this approach, such as the effluent and the recycle flow rates. An improved tank geometry was devised from the results of these studies, achieving a better performance.

Background and relevance

Mathematical models are becoming nowadays a very useful tool for the optimum design of wastewater treatment technologies. Specifically for DAF technology, where variables like tank geometry, flow of water or formation and behavior of bubbles and aggregates affect the flotation process, CFD models provides an optimum solution for the analysis of the process and the design of new units. When constructing a DAF model, phenomena occurring at microscopic level: aggregates formation and fate, and at macroscopic level: flow pattern in the tank, must be considered. However, a CFD model considering three phases (water, air bubbles and aggregates) and interactions between micro-bubbles and flocs forming aggregates is not a straightforward task due to the complexity and uncertainty of all these phenomena and the current computational capacity limitations. For this, as an alternative to the advanced modeling approaches proposed in the literature (e.g. Bondelind et al., 2013), this work presents a CFD model for DAF units that, based on a simplified approach, allows for a realistic description of all relevant phenomena in these units reducing considerably computational costs and consequently, constituting a very suitable design tool for the water engineering companies involved.

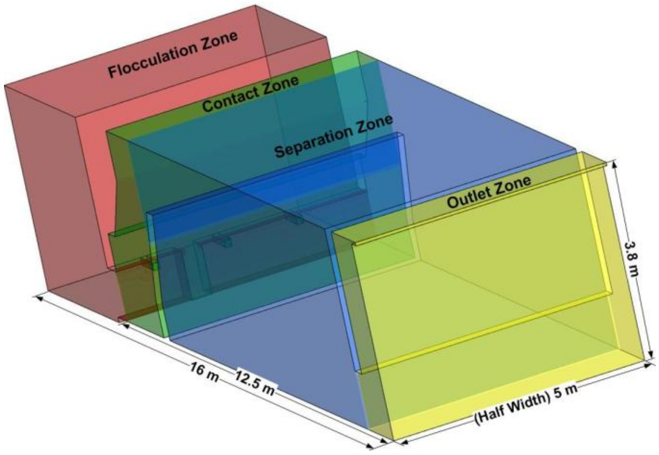

Figure 1. DAF Tank dimensions and zones.

The DAF tank considered in the study is a preliminary design of Tecexsa, an EGILE's company (Spain). The main dimensions of the tank are 16 m (long) x 10 m (wide) x 3.8 m (water height), and it is divided in four zones, namely flocculation zone, contact zone, separation zone and outlet zone (Figure 1). Six injectors located in the contact zone introduce a mixture of recycled water and micro-bubbles. The nominal operating conditions are considered to be an effluent flow rate of 1,000 m^3/h, along with a 121 m^3/h recycle flow rate (12.1%) that carries a 12.3 m^3/h flow rate of air bubbles with a mean diameter of 50 μm.

The flow within the tank has been simulated in ANSYS Fluent V14.5 using the two-phase Euler-Euler multiphase model. The formation and fate of aggregates has been described in a simplified way by just providing the buoyancy characteristics of the aggregates to the bubbles in a two-phase simulation. The characteristics of the aggregates (i.e. mean density and size) and their rise velocity are calculated from the model proposed by Haarhoff and Edzwald (2001), and the diameter of the bubbles is increased to provide them the same rise velocity as the aggregates. The diameter is linearly increased in the contact zone and maintained constant within the separation and outlet zones (Figure 2).

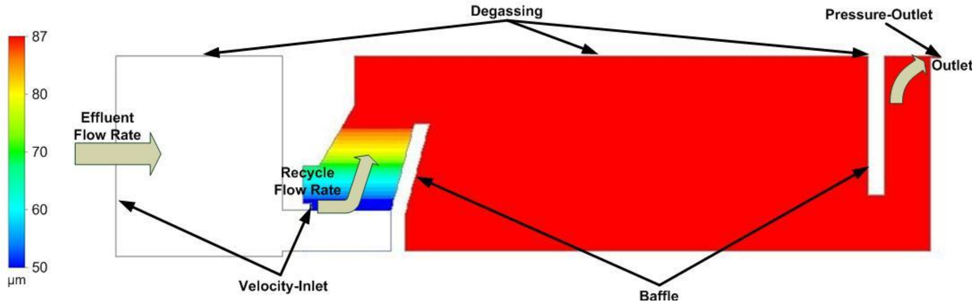

Figure 2. Boundary Conditions and Bubble Sizes.

For this work, flocs of 100 μm with a density of 1,020 kg/m^3 have been assumed, so that aggregates of 135.72 μm with a mean density of 408.73 kg/m^3 would be produced, assuming that 12 bubbles (ρ=1.217 kg/m^3) are added to each floc. These aggregates have the same rise velocity than an 87 μm bubble with a density of ρ=1.217 kg/m^3, thus the diameter of the bubbles is increased from 50 μm to 87 μm in the contact zone (Figure 2).

Results

Different parameters were assessed in order to analyze and discuss the performance of the DAF tank, namely the White Water Level (WWL) and the average vorticity, both proposed by Amato and Wicks (2009), and the percentage of air escaped through the outlet. The WWL represents the boundary between the white water (water, bubbles and aggregates) and the clean water, and it is measured from the bottom of the DAF tank. Moreover, the flow structure was analyzed by the established principles of stratified and shortcircuit flows that were defined by Lundh et al., (2002). The values of these parameters are shown in Table 1 for all the cases studied.

First of all, the simulation for nominal operating conditions ($Q_{effluent}$=1000 m^3/h and 12.1 % of recycle flow rate) shows that the WWL is at the same height as the bottom of the outlet baffle (Figure 3-A). Such position of the WWL leads to a small percentage of air escaped through the outlet (Table 1). Moreover, the flow shows a stratified structure, the velocity vectors being horizontal at the top of the separation zone and taking a downward component below, until they direct to the outlet. However, there exists a shortcircuit flow on the top of the baffle that separates the contact and the separation zones, and a part of the flow goes back from the separation zone into the contact zone (Figure 3-A). This is an undesirable characteristic that should be avoided.

The recycle flow rate was varied from 12.1% to 20% (Figure 3-B) leading to a lower WWL and a higher percentage of air escaped through the outlet (Table 1). The mean vorticity, however, does not change appreciably with this variation. Concerning the flow structure, the flow shows a stratified structure and the previous shortcircuit seen in Figure 3-A disappears.

For an effluent flow rate of 500 m^3/h (Figure 3-C) the aggregates float more easily because of the lower flow velocities. As a result, the WWL is very high and the percentage of air escaped through the outlet is very low (Table 1). In the case of increasing the effluent flow rate up to 2250 m^3/h (Figure 3-D), the WWL is very low and as a consequence the percentage of air escaped through the outlet is very high (Table 1). This poor performance is caused by the significant shortcircuit flow that appears after the contact zone.

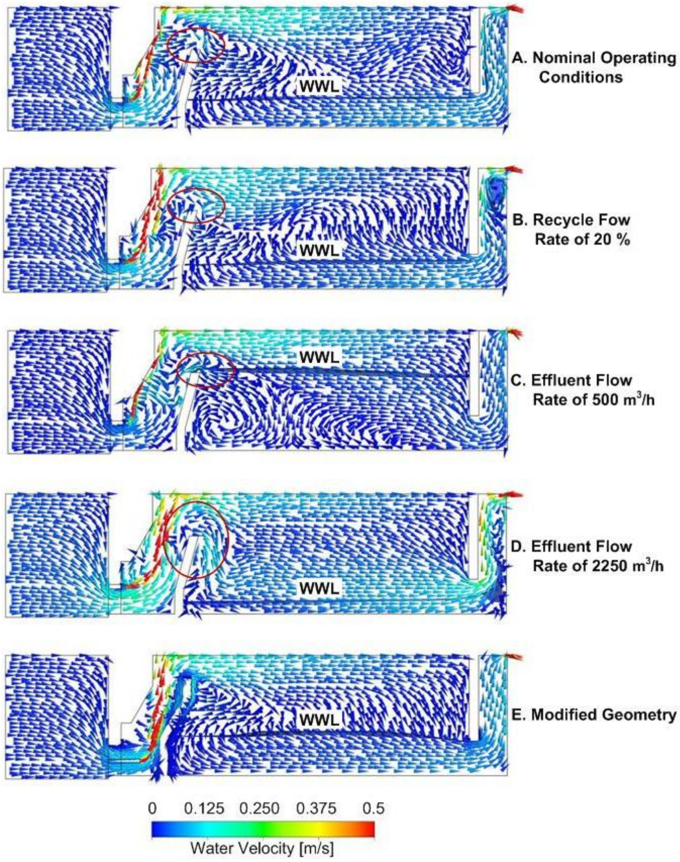

Figure 3. Water Velocity Vectors and White Water Level.

Table 1. Main results of the simulations.

Case	WWL (m)	Air Escaped through Outlet	Vorticity (s^{-1})
A	0.94	2.1%	0.0769
B	0.86	6.3%	0.0755
C	2.51	Negligible	0.0622
D	0.42	29%	0.108
E	1.3	0.06%	0.0672

Finally, the geometry was changed in order to avoid any shortcircuit between the contact and the separation zones, aiming at improving the performance of the DAF tank. The dimensions and shape of the baffle that divides both zones were changed, and the injectors were located lower and more forward (Figure 3-E). As a result of these changes, the WWL is slightly higher than the nominal geometry and the percentage of air escape at the outlet decreases significantly. It is worth mentioning that the mean vorticity in the tank is also lower than in the original geometry (Table 1). Moreover, the shortcircuit that appeared at the top of the baffle in the original geometry disappears and a fully stratified flow structure is obtained.

Discussion

A CFD model-based approach for design and operation of DAF tanks has been presented. Although the presented results must be validated experimentally, they are in close agreement with the experience of TECEXSA, who are using this approach for designing DAF tanks.

References

Amato, T. and Wicks, J. (2009). Dissolved Air Flotation and pontential clarified water quality based on Computational Fluid Dynamics Modelling. AWWA Water Quality Technology Conference, Seattle, 2009.

Bondelind, M., Sasic, S. and Bergdahl, L. (2013) A model to estimate the size of aggregates formed in a Dissolved Air Flotation unit. Applied mathematical Modelling 37, 3036-3047.

Haarhoff, J. and Edzwald, J.K: (2001) Modelling of floc-bubble aggregate rise rates in dissolved air flotation. Water Sci. and Technol. 43 (8) 175-184.

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 Research 36, 1585-1595.