Journal of Water Supply: Research and Technology—AQUA | 58.3 | 2009

A CFD methodology for the design of rectangular sedimentation tanks in potable water treatment plants

Ghawi A. Hadi and Jozef Kriš

ABSTRACT

Several modifications with different construction costs were proposed to enhance the performance in large rectangular sedimentation tanks. A two-dimensional fully mass conservative sedimentation model, based on modern computational fluid dynamics theory, was applied to evaluate proposed tank modifications of the existing and modified sedimentation tanks. The usually unknown and difficult to be measured particle density is found by matching the theoretical to the easily measured experimental total settling efficiency. The proposed strategy is computationally much more efficient than the corresponding strategies used for the simulation of wastewater treatment. It is noteworthy that even small differences in the particle velocity can cause large changes in the percent of settled particles; in this work, the overall solids removal efficiency increased when using the modifications from 84.4 and 98.8%, leading to a reduction of the effluent solids concentration of approximately 85%. The comparison of model predicted results with the field data collected in tanks before and after modifications shows very good agreement.

Key words | baffle, computational fluid dynamics, numerical modeling, sedimentation tank, water treatment

Ghawi A. Hadi (corresponding author)
Jozef Kriš
Department of Sanitary and Environmental Engineering,
Faculty of Civil Engineering,
Slovak University of Technology,
Bratislava Radlinského 11,
813 68 Bratislava 15,
Slovakia
Tel.: +4212 5927 4615
Fax: +4212 5292 1184
E-mail: *hadi.ghawi@stuba.sk*

NOMENCLATURE

F	fractal dimension (m)
S_{eff}	effective resistance coefficient (-)
K	effectiveness of particle settling (%)
n_i	mass fraction (-)
i	particle size group (-)

INTRODUCTION

The purpose of the sedimentation tank is to allow particles sufficient time to settle out of the water. Over the last two decades, there has been a large growth in the application of Computational Fluid Dynamics (CFD) to the design of facilities. CFD has many advantages over traditional modelling approaches as it is a low-cost, high-speed technique for evaluating engineering systems that are difficult to simulate in a laboratory or under field conditions. CFD is able to yield a "virtual prototype" and a good example of this is in its application to the design of compact and more efficient sedimentation tanks for traditional water treatment plants. CFD can capture the three-dimensional fluid flow inside a tank and thus help to minimize turbulence and optimize solids separation.

Many researchers have used Computational Fluid Dynamics (CFD) simulations to describe water flow and solids removal in settling tanks for sewage water treatment. However, works in CFD modelling of rectangular sedimentation tanks for potable water treatment have not been found in the literature. Ghawi & Kris (2007*a*,*b*) investigated the effect of various solids and hydraulic loads on sedimentation tank performance in water treatment plants. Moreover, the physical characteristics of the flocs may not be such significant parameters in the flow field of sedimentation tanks for potable water, due to the much lower solids

doi: 10.2166/aqua.2009.027

concentrations and greater particle size distributions than those encountered in wastewater treatment.

The aim of this project was originally stated as to improve the operation and performance of horizontal sedimentation tanks in the Holic water treatment plant (WTP) which have been identified as operating poorly, by predicting the existing flow and flocculent concentration distribution of the sedimentation tank by means of CFD techniques.

The objective of this work was to develop a new CFD methodology for the analysis of sediment transport for multiple particle sizes in sedimentation tanks of potable water treatment plants (the place is Slovakia in the Holic WTP), and to improve the operation and performance of horizontal sedimentation tanks in the Holic WTP which have been identified as operating poorly, by analyzing and modeling the two-phase flow regime found in settling tanks and comparing the model with experimental data. The two phases present are water, the continuous medium, and floc concentration, the dispersed phase. For this study the flow field is considered to be isothermal, incompressible and without phase change. The CFD package FLUENT 6.3.26 was used for the case study of the effect of adding energy dissipation baffles, a flocculation zone, perforated and non-perforated baffles, and modifications to the launders and on the efficiency of solids removal.

MATERIALS AND METHODS

Flow solver

The computational fluid dynamics code FLUENT 6.3.26 has been used to carry out the simulations. Either a Eulerian or a Lagrangian approach can be adopted to model the particulate phase. In the literature, Eulerian applications are used for almost all diffusion-dominated problems, so strictly speaking they are only suitable for gas or ultrafine particle study (De Clercq & Vanrolleghem 2002). Due to their versatile capabilities, approaches based on the Lagrangian method have been applied extensively for many two-phase flow problems.

In these approaches, the fluid is treated as a continuum and the discrete phase is treated in a natural Lagrangian manner, which may or may not have any coupling effect with fluid momentum. De Clercq & Vanrolleghem (2002) mentioned that the Lagrangian model should not be applied whenever the particle volume fraction exceeds 10 – 12%.

The trajectories of individual particles through the continuum fluid using the Lagrangian approach are calculated in FLUENT by the Discrete Phase Model (DPM). The particle mass loading in a sedimentation tank for potable water treatment is typically small, and therefore it can be safely assumed that the presence of particles does not affect the flow field (one-way coupling).

In addition, the volume fraction of the particles in the tank is of the order of 10^{-4}. The turbulent coagulation is well known to be proportional to this volume fraction, so it can be ignored under the present conditions. Also, the coagulation due to differential settling can be ignored due to the relatively low settling velocities resulting from the low densities of the flocs.

The settling velocity hindering is insignificant for these levels of solids volume fraction as can be shown by employing the corresponding theories. Moreover, Lyn *et al.* (1992), based on model observations, concluded that for conditions of relatively small particle concentrations in sedimentation tanks, the coalescence of the flocs does not affect the flow field and the effects on the concentration field and the removal efficiency may be of secondary importance. Finally, decreases in the size range relevant to primary separators do not suffer breakage (Wilkinson *et al.* 2000).

The final system of particle conservation equations is a linear one, so the superposition principle can be invoked to estimate the total settling efficiency. The inlet particle size range is divided into classes with the medium size of each class assumed as its characteristic. Then independent simulations are conducted for monodisperse particles in the feed using the individual particle sizes every time. The overall settling efficiency can be found by adding appropriately the efficiency for each particle size.

Tracks are computed by integrating the drag, gravitational and inertial forces acting on particles in a Lagrangian frame of reference. The dispersion of particles due to turbulence is modeled using a stochastic discrete-particle approach. The trajectory equations for individual particles are integrated using the instantaneous fluid velocity along the particle path during the integration. By computing the trajectory in this manner for a sufficient number of representative particles, the random effects of turbulence on particle dispersion may be accounted for.

Sedimentation tank

A full-scale rectangular sedimentation tank was investigated, similar to those used in the potable water treatment plant of the city of Holic. There are two rectangular tanks at the Holic water treatment plant. The plant receives raw water from groundwater and its capacity is around 200 l/s. The employed processes include aeration, coagulation–flocculation, sedimentation, filtration through sand and chlorination. The volume of the sedimentation tank is 500 m^3 (Figure 1). The goal when designing this baffle is to achieve some head loss while keeping the velocity gradients through the ports equal to (or lower than) the velocity gradient in the end of the flocculator, so as to not break up the flocs. In each tank there is one effluent weir, which goes along the longitude direction and is distributed over a range of 2 m from the side wall. The multiple scrapers driven by a chain move toward the tank upstream hopper to collect the settled sludge into the sludge hopper located at the beginning of the tank.

Longitudinal velocity was determined by means of submerged drifters, a method that involves measuring the time taken for a drifter at the measurement depth to traverse a given distance from which the velocity could be calculated. Accuracy is given to ± 2 mm/s.

Solids concentration was measured by an ultrasound probe. Interference with concentration measurements occurred when the probe was placed within 0.2 m of the walls or sludge scraper; hence no readings were taken closer than this distance. Also these measurements were limited to an upper solids fraction of about 0.07, around 1.3 times the inlet solids fraction.

Particle size distribution—experimental determination

Samples of incoming and effluent suspensions were taken and analyzed for particle size distribution using the laser diffraction technique, with a Malvern Mastersizer 2000 analyzer. The location of sampling is very important and depends on the goal of the study. The numerical model is compared with data obtained in a settling tank. In this study, mean velocity and solids fraction concentration were measured at a number of stations along the length and across the width of the tank.

The time taken to gather all the data for one inlet condition was around 3 h, so no definitive time was given for the measurements at each station. Because of this, the numerical simulation was compared with the experimental data at a time when the settled sludge layer in the simulation was at approximately the same height as that found in the experiment. At this time, the flow field above the settled sludge layer should be similar in both the experiment and the simulation.

The influence of particle structure

The settling velocity of an impermeable spherical particle can be predicted from Stokes' law. However, the aggregates in the water are not only porous but it is well known that they have quite irregular shapes with spatially varying porosity. The description of the aggregates as fractal objects is the best possible one-parameter description of their complex structure, so it has been extensively used in the literature. The well-known fractal dimension, F, is a

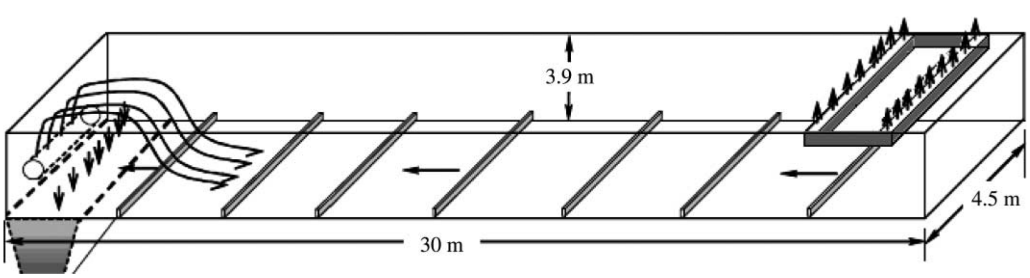

Figure 1 | Schematic representation of the Holic sedimentation tank in 3D.

quantitative measure of how primary particles occupy the floc interior space. But the settling velocity of the aggregate depends on its structure both through its effective density and its drag coefficient. These variables must be independently estimated for the aggregate shapes instead of the settling velocity because settling velocity cannot be directly entered into the CFD code. As regards the drag coefficient of fractal aggregates, ample information can be found in the literature; from simulation of the flow inside reconstructed flocs using the FLUENT code to purely empirical relations. According to Gmachowski (2005), the ratio of the resistance experienced by a floc to that of an equivalent solid sphere (S_{eff}) can be expressed as follows:

$$S_{\mathrm{eff}} = \sqrt{1.56 - \left(1.728 - \frac{F}{2}\right)^2} - 0.288 \tag{1}$$

where S_{eff} is the effective resistance coefficient and F is the fractal dimension. It is found that aggregates generated in water treatment processes exhibit a fractal dimension ranging between 2.2–2.6 (Lee *et al.* 1996), so the resistance coefficient, S_{eff}, varies between 0.84 – 0.94. This estimation agrees also with the theoretical results for the drag coefficient of fractal aggregates given by Vanni (2000) who solved the problem in the limit of zero Reynolds number.

Contrary to the drag coefficient, the effective density cannot be estimated at all. Even if the structure (and the porosity) of the aggregate is known, the intrinsic density of the primary particles is not known. As the settling velocity is not so sensitive to S_{eff} (the sensitivity with respect to effective density is much larger, since the difference between effective and water density determines the settling velocity), the resistance coefficient was fixed at 0.91 and the apparent density was then estimated as 1,065 $\mathrm{kg\,m^{-3}}$ by requiring the final computed settling effectiveness to coincide with the measured settling effectiveness of 91%. This is a typical effective density value for the aggregates met in water treatment applications (Deininger *et al.* 1998).

Simulation

To limit computational power requirements, the rectangular settling tank was modelled in 2D. The major assumption in the development of the model is that the flow field is the same for all positions; therefore, a 2D geometry can be used to properly simulate the general features of the hydrodynamic processes in the tank. As a first step, a mesh was generated across the sedimentation tank. The segregated solution algorithm was selected. The $k-\varepsilon$ turbulence model was used to account for turbulence, since this model is meant to describe better low Reynolds numbers flows such as the one inside our sedimentation tank. The discretization schemes used were the simple for the pressure, the PISO for the pressure–velocity coupling and the second-order upwind for the momentum, the turbulence energy and the specific dissipation. Rodi (1993) pointed out that, for real settling tanks, the walls can be considered as being smooth due to the prevailing low velocities and the correspondingly large viscous layer. Consequently, the standard wall functions as proposed by Launder & Spalding (1974) were used. The water-free surface was modeled as a fixed surface; this plane of symmetry was characterized by zero normal gradients for all variables.

As a first step, the fluid mechanics problem was solved in the absence of particles to find the steady state flow-field. The converged solution was defined as the solution for which the normalized residual for all variables was less than 10^{-6}. In addition, the convergence was checked from the outflow rate calculated at each iteration of the run. The convergence was achieved when the flow rate calculated to exit the tank no longer changed.

The settling tank was simulated for a specific set of conditions used in the Holic treatment plants for which the particle size distribution at the inlet and outlet and the total settling efficiency has been experimentally measured. The inlet was specified as a plug flow of water at 0.035 $\mathrm{m\,s^{-1}}$ and 0.065 $\mathrm{m\,s^{-1}}$, respectively, whereas the inlet turbulence intensity was set at 4.6%. The outlet was specified as a constant pressure outlet with a turbulence intensity of 6.0%. The water flow rate was 0.06 $\mathrm{m^3\,s^{-1}}$ and 0.08 $\mathrm{m^3\,s^{-1}}$, respectively.

For simulation purposes, the range of the suspended solids was divided into 13 distinct classes of particles based on the discretization of the measured size distribution. The number of classes was selected in order to combine the solution accuracy with short computing time. Two other numbers, 6 and 15, were tested. While the predictions obtained using 6 classes of particles were found to be different from those resulting from 13 classes, the difference between the predictions made by the 13 and 15 classes were insignificant. Therefore, a number of 13 classes were selected as suitable ones. Within each class the particle diameter is assumed to be constant (Table 1). As can be seen in Table 1, the range of particle size is narrower for classes that are expected to have lower settling rates.

The procedure used to determine the overall settling effectiveness, K, was based on the calculation of the percent of solids settled for each particle size class, K_i:

$$K = \frac{\sum_{i=1}^{K} (n_i K_i)}{\sum_{i=1}^{K} n_i} \quad (2)$$

where K is the effectiveness of particle settling and n_i is the mass fraction. The settling efficiency for each particle size class was calculated after the conclusion of the 13 different sedimentation simulations for the standard and the modified tank, respectively. In each run, only one particle size class was taken into account; all injected particles were considered to have the same diameter corresponding to the so-called pivot particle size and assumed to be the average of the lower and upper diameters of the class.

The effectiveness of particle settling is estimated as the percentage of solids settled over the rate of solids introduced from the inlet.

Table 1 | Classes of particles used to account for the total suspended solids in the STs in Holic and Hrinova STs

Class	Range of particle sizes (μm)	Mass fraction
1	10–30	0.025
2	30–70	0.027
3	70–90	0.039
4	90–150	0.066
5	150–190	0.095
6	190–210	0.115
7	210–290	0.126
8	290–410	0.124
9	410–490	0.113
10	490–610	0.101
11	610–690	0.077
12	690–820	0.057
13	820–900	0.040

In this way, to predict the overall percent solids removal efficiency one needs to know only the particle size distribution in the influent. With this knowledge and the percentages K_i calculated from the 13 simulations, the sedimentation efficiency could be calculated for any different particle size distribution in the influent.

RESULTS AND DISCUSSIONS

Simulation of existing tank

A rectangular sedimentation tank in Slovakia water treatment plants was selected to demonstrate the response of rectangular tanks to different internal geometries. This case is based on the Holic settling tanks describe by Ghawi & Kris (2007*c*, 2008). This tank was selected because performance data are available for model calibration and because it represents a marginal performance case.

As shown in Figure 2(a, b), the predicted hydraulic regime typically consists of the upward inlet jet, the influent density waterfall, a bottom density current and a strong surface reverse flow in the absence of proper baffling. For a case with a thick sludge blanket, the simulated velocity field showed that the bottom density current deflects upward while near the tank bottom a strong reverse sludge flow appears. According to both the field observations and the modeling of the existing process, each of the following reasons (or combination of them) may cause the Sedimentation Tank (ST) problems, i.e. the flocculant solids blowing out:

1. The location of the existing weir (distributed in a range of 1 m at the very downstream end of the ST) can cause very strong upward currents, which could be one of the major reasons that the flocculant solids were blowing out around the effluent area.
2. The strong upward flow is not only related to the small area the effluent flow passes through but also to the rebound effect between the ST bottom density current and the downstream wall. The “rebound” phenomenon has been observed and reported by many operators as well as field investigators, especially in ST with small amounts of sludge inventory. A reasonable amount of

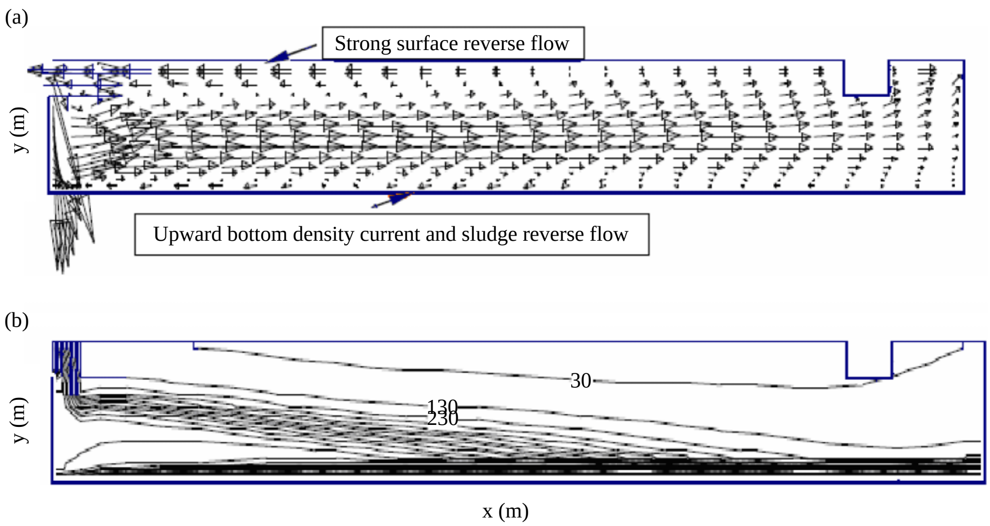

Figure 2 | Flow pattern (a) and sludge blanket (b) (in a vertical section along tank central axis) in existing tank.

sludge inventory can help dissipate the kinetic energy of the bottom density current.

In the existing operation, the bottom density current must be fairly strong due to the lack of proper baffling and the shortage of sludge inventory in the tank.

Proposed tank modifications

The proposed modifications include: energy dissipation baffles, a flocculation zone, perforated and non-perforated baffles, modifications to the launders and modifications of the sludge withdrawal facility. In the original project, a total of five proposed modifications and combinations of them were tested. The following major modifications adopted in final tank construction are presented as:

- Modification 1: inlet flocculation baffle, the distance from tank influent to the baffle = 6 m and the baffle depth = 2.5 m;
- Modification 2: a perforated baffle between bays A and B with slot space of 54% of flow cross-sectional area;
- Modification 3: a perforated baffle between bays B and C with slot space of 66% of flow cross-sectional area;
- Modification 4: a conventional baffle between bays A and B with baffle depth of 1.75 m below the surface;
- Modification 5: removing existing surrounding effluent weir and adding 4–5 new launders in bay C (Figure 3). All effluent launders are aligned with the tank longitudinal direction. The launders extend from the end wall to the perforated baffle between B and C.

The predicted flow and solids fields in the tank with modifications 1, 2, 3 and 5 are presented in Figure 4(a, b). The flow pattern shows that influent density waterfall and surface reverse flow were significantly reduced by the three baffles. The flocculation baffle eliminates most of the entrainment flow from the surface clear water layers into the influent density flow: thus both the surface return flow along the entire tank surface and the bottom density current are substantially reduced (see Figures 2(a) and 4(a)).

The distribution of the sludge blanket among the three bays has been significantly changed by using two perforated baffles (see Figures 2(b) and 4(b)). In the flocculation zone relatively minor solids compression takes place in the local sludge blanket. The highest sludge blanket occurs in bay A due to the high resistance of the perforated baffle A/B. The lowest sludge blanket appears in bay C. The difference of the sludge blanket level between bays B and C is relatively small due to the lower resistance of perforated baffle B/C. The predicted flow pattern and solids field in Figure 4(a, b) show that solids spill gently over the baffle slots, at a lower velocity and potential energy head than that in the upstream bays.

Besides the evaluation of the impact of modifications on the sedimentation tank performance, the sedimentation

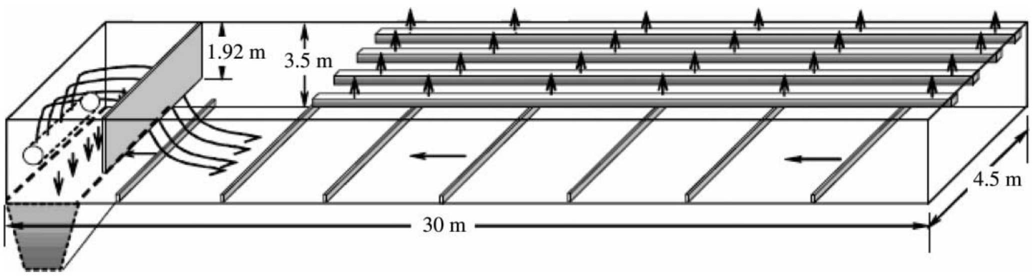

Figure 3 | Tank with baffle and launder Modification 5.

tank model also helps us to implement cost-effective sedimentation tank modifications in this study. The model prediction shows that an optimized baffle design (Modifcations 1, 2 and 3), which needs about half a million dollars in construction cost for all tanks, can give more than 85% improvement of effluent quality. However, the construction costs were about half a million dollars for all tanks due to the necessity of a large amount steel in the new launder-supporting structure. The baffle modifications show the dominant improvement of tank hydraulics behavior, especially in tanks running with a normal sludge blanket level.

Figure 5 presents the predicted percents of solids settled for different tested particle size classes. As can be inferred, the theoretical settling efficiency tends to nonzero (in fact, relatively large) values as the particle size tends to zero. This is due to the combined effect of convection (fluid velocity towards the bottom of the tank, turbulent diffusivity which is independent of the particle size and the perfect sink boundary condition). In practice it is expected that the settling efficiency decreases as the particle size decreases, going to a zero (or close to zero) value for Brownian particles. Although this inconsistency is not exhibited in the case studied here due to the relatively large particle sizes of the feed, it must be considered for the sake of completeness of the simulation procedure. The easiest way to accommodate the realistic behavior of a decreasing settling efficiency as particle size decreases is by incorporating a particle-size-dependent trapping probability in the Lagrangian code. This probability should depend on the interparticle (particle deposits) interactions, turbulent diffusivity and gravity. As the particle size decreases the effect of gravity decreases, leading the probability from unity to the inverse of the stability ratio well known in the studies of small particle

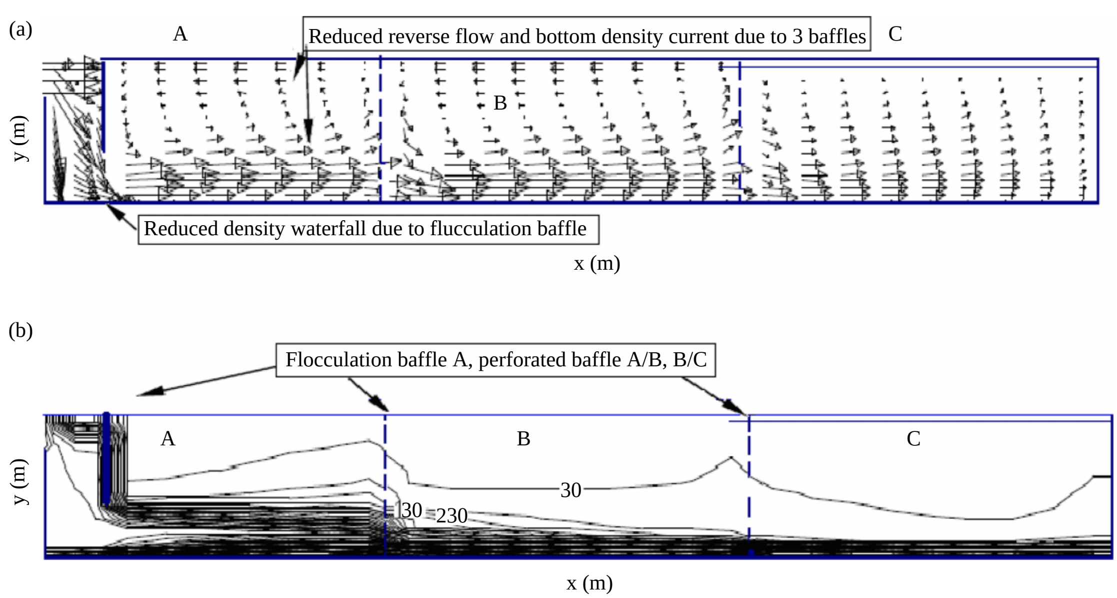

Figure 4 | Flow pattern (a) and sludge blanket (b) (in a vertical section along tank central axis) in tank with Modifications 1, 2, 3 and 5.

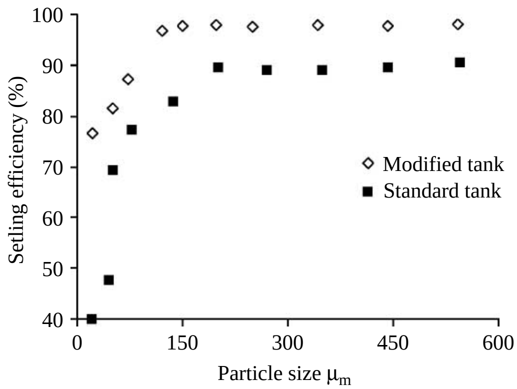

Figure 5 | Predicated percents of solids settled for each particle size class.

deposition from turbulent flows (Kostoglou & Karabelas 1998). Nevertheless, despite the aforementioned improvement, the present CFD model provides a good overall description of the system behavior.

The percentages presented in Figure 5 result in an overall settling efficiency of 84.4 and 98.8% for the existing and modified tank, respectively. As can be seen, the model predicts a highly distinct concentration for different classes of particle: lower removal rates for the smallest and higher removal rates for the heaviest particles.

Therefore, the improvement in the overall efficiency of solids removal is only achieved by improving the settling of particles with lower settling velocities (classes 1–5 (Table 1)). This observation is similar to that obtained by Huggins *et al.* (2005), who used a CFD model to evaluate the impact of potential raceway design modifications on the in-raceway settling of solids.

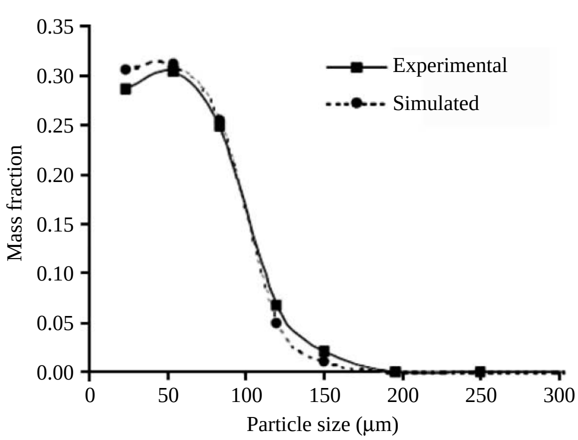

Figure 6 | Simulated and experimental particle size distribution in the effluent of the existing tank.

Although the increase in the overall effectiveness seems small it corresponds to an estimated reduction in the solids exiting the tanks of approximately 520 kg d^{-1} or to a reduction of about 85% of the solids in classes 1–13 (Table 1) that exit the tank. These values are greater than those reported by other researchers. Huggins *et al.* (2005), who tested a number of potential raceway design modifications, noticed that by adding baffles the overall percent solids removal efficiency increased from 81.8 to 91.1%, resulting in a reduction of the effluent solids of approximately 51%. Crosby (1984) used an additional baffle at mid-radius extending from the floor upwards to mid-depth and observed a reduction of 38% in effluent concentration.

Validation of solids concentration profiles

As far as the CFD model validity is concerned, Figure 6 presents a comparison between the experimentally measured and the simulated values of the floc size distribution in the effluent of the existing tanks. Apparently, there is a good agreement between measured and predicted values.

CONCLUSION

This work deals with the development of a specialized strategy for the simulation of the treatment of potable water in sedimentation tanks. The strategy is based on the CFD code FLUENT and exploits several specific aspects of the potable water application (low solids mass and volume fraction) to derive a computational tool computationally much more efficient (due to the independent handling of flow fields and different particle classes) than the corresponding tools employed to simulate wastewater settling tanks. The present code is modified based on data from a real sedimentation tank.

The 2D fully mass conservative ST model is applied to predict the tank performance in the existing STs and the STs with proposed modifications at the Holic water treatment plants. The existing Holic STs suffered from relatively poor hydraulic performance which typically occurs in tanks without proper baffling. The unfavorable hydraulic regime includes strong turbulence, a high influent potential energy and a strong density current due to excessive flow entrainment. The baffle modifications can considerably reduce the strength of the density flow and increase the solids detention time in the tank; the effluent quality can be improved by more than 85% in some cases. Proper launder modifications can be used to improve local flow pattern near the effluent weir and to re-distribute the effluent flow along the tank longitudinal direction.

ACKNOWLEDGEMENTS

Research Grant VEGA No. 1/3313/06 and project KEGA 3/5125/07 solved at the Department of Sanitary and Environmental Engineering, Faculty of Civil Engineering, Slovak University of Technology, Bratislava have supported this paper.

REFERENCES

Crosby, R. M. 1984 *Hydraulic Characteristics of Activated Sludge-secondary Clarifiers* EPA 600/2-84-131, NTIS no. PB-84-229665. US EPA, Municipal and Environmental Research Laboratories, Cincinnatti, OH.

De Clercq, B. & Vanrolleghem, P. A. 2002 Computational fluid dynamics in wastewater treatment. *Med. Fac. Landb. Toegep. Biol. Wet.* **67**, 15–19.

Deininger, A., Holthausen, E. & Wilderer, P. A. 1998 Velocity and solids distribution in circular secondary clarifiers: full scale measurements and numerical modelling. *Water Res.* **32**(10), 2951–2958.

Gmachowski, L. 2005 Aggregate structure and hydrodynamics of aggregated systems. *Colloid Surf. A: Physicochem. Eng. Aspects* **255**, 105–110.

Hadi, G. A. & Kris, J. 2007*a* Design and optimization of sedimentation tank in Slovakia with CFD modeling. In: *Proc. 10th International Symposium on Water Management and Hydraulic Engineering 2007 with Special Emphasis on the Impact of Hydraulic Engineering Construction on the Environment, Šibenik, Croatia, 4–9 September*. Zagreb Faculty of Civil Engineering, Zagreb, CD.

Hadi, G. A. & Kris, J. 2007*b* A numerical model of flow in sedimentation tanks in Slovakia. In: *Proc. 3rd International PhD Symposium in Engineering, Hungary University of Pollack Mihály Faculty of Engineering Pécs, Hungary, 25–26 October*. *Pollack Periodica*, **3**(2), 59–73.

Hadi, G. A. & Kris, J. 2007*c* Improved, modeling, simulation and operational parameters of settling tank. In: *Proc. 6th International Conference of PhD Students, University of Miskolc, Hungary, 12–18 August*. pp. 69–75.

Hadi, G. A. & Kris, J. 2008 Design and optimization of settling tanks performances in Slovakia. In: *Proc. 20th National and 8th International Scientific and Technical Conference Water Supply and Water Quality Water, Poland, POZNAN-GNIEZNO, 15–18 June*. Part 1, pp 425–438.

Huggins, D. L., Piedrahita, R. H. & Rumsey, T. 2005 Use of computational fluid dynamics (CFD) for aquaculture raceway design to increase settling effectiveness. *Aquacult. Eng.* **33**, 167–180.

Kostoglou, M. & Karabelas, A. J. 1998 Comprehensive modeling of precipitation and fouling in turbulent pipe flow. *Indus. Eng. Chem. Res.* **37**, 1536–1550.

Launder, M. & Spalding, D. 1974 The numerical computation of turbulent flows. *Comput. Meth. Appl. Mech. Eng.* **3**, 269–289.

Lee, D. J., Chen, G. W., Liao, Y. C. & Hsieh, C. C. 1996 On the free-settling test for estimating activated sludge floc density. *Water Res.* **30**, 541–550.

Lyn, D. A., Stamou, A. & Rodi, W. 1992 Density currents and shear induced flocculation in sedimentation tanks. *J. Hydraul. Eng.* **118**, 849–867.

Rodi, W. 1993 *Turbulence Models and Their Application in Hydraulics–A State-of-the-art Review* 3rd edition. IAHR Monograph Series, A.A. Balkema, Rotterdam.

Vanni, M. 2000 Creeping flow over spherical permeable aggregates. *Chem. Eng. Sci.* **55**, 685–698.

Wilkinson, D., Waldie, B., Mohamad Nor, M. I. & Lee, H. Y. 2000 Baffle plate configurations to enhance separation in horizontal primary separators. *J. Chem. Eng.* **77**(3), 221–226.

First received 17 March 2008; accepted in revised form 12 September 2008. Available online February 2009