ELSEVIER

Chemical Engineering Journal 140 (2008) 110–121

Chemical Engineering Journal

www.elsevier.com/locate/cej

A CFD methodology for the design of sedimentation tanks in potable water treatment Case study: The influence of a feed flow control baffle

Athanasia M. Goula, Margaritis Kostoglou, Thodoris D. Karapantsios *, Anastasios I. Zouboulis

Division of Chemical Technology, School of Chemistry, Aristotle University, University Box 116, 541 24 Thessaloniki, Greece

Received 14 June 2007; received in revised form 13 September 2007; accepted 14 September 2007

Abstract

Computational fluid dynamics simulations are employed to assess the effect of adding a vertical baffle at the feed section of a full-scale sedimentation tank for the improvement of solids settling in potable water treatment. A general CFD-based simulation strategy is developed based on the specific features and conditions met in practice for potable water treatment. The linearity of the particle conservation equations allows separate calculations for each particle size class – but performed for all classes of interest – leading to the uncoupling of the CFD problem from a particular inlet particle size distribution. 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. This work compares simulations from a standard and a baffle-equipped tank. It is found that the baffle decreases the inlet recirculation zone and enhances the settling of solids by directing them towards the bottom of the tank with high velocities. 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 baffle from 90.4 to 98.6% leading to a reduction of the effluent solids concentration of approximately 85%.

Keywords: Baffle; Computational fluid dynamics; Numerical modelling; Sedimentation tank; Water treatment

1. Introduction

Sedimentation is perhaps the oldest and most common water treatment process. The principle of allowing turbid water to settle before it is drunk can be traced back to ancient times. In modern times a proper understanding of sedimentation tank behavior is essential for proper tank design and operation. Generally, sedimentation tanks are characterized by interesting hydrodynamic phenomena, such as density waterfalls, bottom currents and surface return currents, and are also sensitive to temperature fluctuations and wind effects. On the surface, a sedimentation tank appears to be a simple phase separating device, but down under an intricate balance of forces is present.

Many factors clearly affect the capacity and performance of a sedimentation tank: surface and solids loading rates, tank type, solids removal mechanism, inlet design, weir placement and loading rate, etc. To account for them, present-day designs are typically oversizing the settling tanks. In that way, designers hope to cope with the poor design that is responsible for undesired and unpredictable system disturbances, which may be of hydraulic, biological or physico-chemical origin.

To improve the design of process equipment while avoiding tedious and time consuming experiments computational fluid dynamics (CFD) calculations have been employed during the last decades. Fluid flow patterns inside process equipment may be predicted by solving the partial differential equations that describe the conservation of mass and momentum. The geometry of sedimentation tanks makes analytical solutions of these equations impossible, so usually numerical solutions are implemented using computational fluid dynamics packages. The advent of fast computers has improved the accessibility of CFD, which appears as an effective tool with great potential. Regarding sedimentation tanks, CFD may be used first for optimizing the design and retrofitting to improve effluent quality and underflow solids concentration. Second, it may increase the basic understanding of internal processes and their interactions. This knowledge can again be used for process optimization. The latter concerns the cost-effectiveness of a validated CFD model

* Corresponding author. Tel.: +30 2310 997772; fax: +30 2310 997759.
E-mail address: karapant@chem.auth.gr (T.D. Karapantsios).

1385-8947/$ – see front matter
doi:10.1016/j.cej.2007.09.022

Nomenclature

c	mass fraction
D	fractal dimension (m)
f_{eff}	effective resistance coefficient
i	particle size group
n	effectiveness of particle settling (%)

where simulation results can be seen as numerical experiments and partly replace expensive field experiments [1].

From a hydraulic point of view, a distinction has to be made between primary and secondary settling tanks in terms of density effects. In secondary clarifiers the increased density (due to large particle mass fraction) gives rise to a couple of characteristic flow features such as density waterfall phenomenon near the inlet of the clarifier and solid cascading phenomenon in clarification of suspended solids [2]. In the case of potable water treatment the solid mass fraction is even smaller than that of the primary clarifiers.

Much research has been done on secondary sedimentation tanks for wastewater treatment. Larsen [3] was probably the first who applied a CFD model to several secondary clarifiers and, although his model incorporated several simplifications, he demonstrated the presence of a "density waterfall", which is a phenomenon that causes the incoming fluid to sink to the tank bottom soon after entering. Shamber and Larock [4] used a finite volume method to solve the Navier–Stokes equations, the k–ε model and a solids concentration equation with a settling velocity to model secondary clarifiers. McCorquodale et al. [5] developed a model using a combination of finite element methods (for the stream function) and finite difference methods (for the boundaries). McCorquodale and Zhou [6] investigated the effect of various solids and hydraulic loads on circular clarifier performance, whereas Zhou et al. [7] linked the energy equation with the Navier–Stokes equations to simulate the effect of neutral density and warm water into a model clarifier. Krebs et al. [8] used the Phoenics code to model different inlet arrangements and evaluated the effect of inlet baffle position and depth. Deininger et al. [9] improved the Champion3D numerical model and predicted the velocity and solids distribution in a circular secondary clarifier, whereas Kim et al. [2] have recently performed a numerical simulation in a 2D rectangular coordinate system and an experimental study to figure out the flow characteristics and concentration distribution of a large-scale rectangular final clarifier in wastewater treatment.

With respect to primary sedimentation tank, where the solids concentration is limited and discrete settling prevails, Imam et al. [10] applied a fixed settling velocity and used an averaged particle velocity. Stamou et al. [11] simulated the flow in a primary sedimentation tank using a 2D model in which the momentum and solid concentration equations were solved but not linked to account for buoyancy. Adams and Rodi [12] used the same model as in 1989 and did extensive investigations on the inlet arrangements and the flow through curves. More advanced is the work of Lyn et al. [13] that accounts for flocculation where six different size classes with their respective velocities were considered. Frey [14] used the VEST code to determine the flow pattern in a sedimentation tank. The flow profiles were then used by the TRAPS code to determine particle tracks. Van der Walt [15] used the 3D Flo++ code to determine the sensitivity of a primary sedimentation tank behavior on a number of geometric, fluid and solids transport properties and simulated the existing Vaalkop sedimentation tanks using a 3D pseudo two-phase model demonstrating how the inlet geometry was the main cause of the poor desludging capacity.

Generally, many researchers have used CFD simulations to describe water flow and solids removal in settling tanks for sewage water treatment. However, works in CFD modelling of sedimentation tanks for potable water treatment have not been found in the literature. Moreover, the physical characteristics of the flocs may not be such significant parameters in the flow field of clarifiers for potable water, due to the much lower solids concentrations and greater particle size distributions than those encountered in wastewater treatment.

The objective of this work was to develop a new CFD methodology for the analysis of the sediment transport for multiple particle sizes in full-scale sedimentation tanks of potable water treatment plants. The CFD package FLUENT 6.2.16 was used for the case study of the effect of adding a feed flow control baffle on the efficiency of solids removal. The structure of the present work is the following: Section 2 presents the CFD-based simulation strategy developed with respect to the specific features and conditions of a potable water sedimentation tank. It describes the ways of particle trajectories calculation, based on their small mass loading, and of the handling of the different particles size classes. In addition, the influence of particle structure is discussed and a method for particle density calculation is developed. Section 3 presents the outcomes of the standard and the modified (baffle-added) tank simulations concerning the flow pattern and the solids distribution and discusses the CFD model validity and the influence of the baffle on the solids settling. Finally, conclusions drawn from this study are presented in Section 4.

2. Materials and methods

2.1. Flow solver

The computational fluid dynamics code FLUENT 6.2.16 has been used to carry out the simulations. The code predicts fluid flow by numerically solving the partial differential equations, which describe the conservation of mass and momentum. A grid is placed over the flow region of interest and by applying the conservation of mass and momentum over each cell of the grid sequentially discrete equations are derived. In the case of turbulent flows, the conservation equations are solved to obtain time-averaged information. Since the time-averaged equations contain additional terms, which represent the transport of mass and momentum by turbulence, turbulence models that are based on a combination of empiricism and theoretical considerations are introduced to calculate these quantities from details of the mean flow. Either an Eulerian or a Lagrangian approach can be adopted to model 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 [16]. 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 (particle) phase is treated in a natural Lagrangian manner, which may or may not have any coupling effect with fluid momentum. De Clercq and Vanrolleghem [16] 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). This means that the fluid mechanics problem can be solved in the absence of particles to find the steady state flow field. Then the particles, whose density and size could be assigned at will, are released from the inlet and are tracked along their trajectories. 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 by the low densities of the flocs. The settling velocity hindering is insignificant for these levels of solids volume fraction as it can be shown by employing the corresponding theories [17]. Moreover, Lyn et al. [13], based on model observations, concluded that for conditions of relatively small particles concentrations in sedimentation tanks, the flocs coalescence do not affect the flow field and the effects on the concentration field and the removal efficiency may be of secondary importance. Finally, drops in the size range relevant to primary separators do not suffer breakage [18].

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 in classes with the medium size of each class assumed as its characteristic (pivot). Then independent simulations are conducted for monodisperse particles in the feed using every time the individual pivot sizes. 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.

2.2. Sedimentation tank

A full-scale circular sedimentation tank was investigated, similar to those used in the potable water treatment plant of the city of Thessaloniki. The plant receives raw water from Aliakmon river and its capacity is around 150,000 $m^3\,day^{-1}$. The employed processes include pro-ozonation, coagulation–flocculation, sedimentation, sludge thickening, filtration through sand and active carbon, ozonation and chlorination. The sedimentation tank, with a volume of 2960 m^3, is centre-fed with a peripheral weir. The bottom floors have a steep slope of 12.5° and a blade scraper pushes the sludge towards a central conical sludge hopper. Two tank configurations have been considered. One with only a small vertical baffle to guide the feed of the tank henceforth referred to as standard tank (Fig. 1a). And another where the small baffle is extended by an inclined and a second vertical section, altogether meant to guide the fluid significantly deeper inside the tank, henceforth referred to as modified tank (Fig. 1b).

2.3. 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. When samples are taken at the tank overflow exit, care was exercised in order to sample from a well-mixed and representative location. Sampling took place during at least twice the theoretical residence time. If the system is unclear or there are known dead spaces, it might be considered to prolong the measurement campaign to capture the complete hydraulic behavior.

Drawbacks of the laser diffraction technique are the assumption of sphericity of particles in the optical model and the required dilution step to avoid multiple scattering, because this is

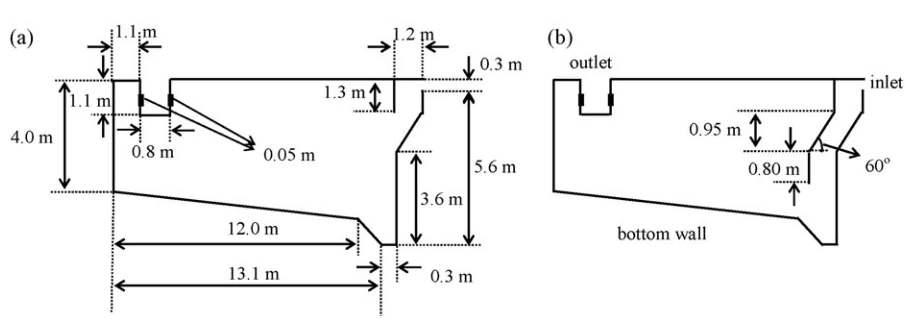

Fig. 1. Schematic representation of the standard (a) and the modified (b) simulated sedimentation tank.

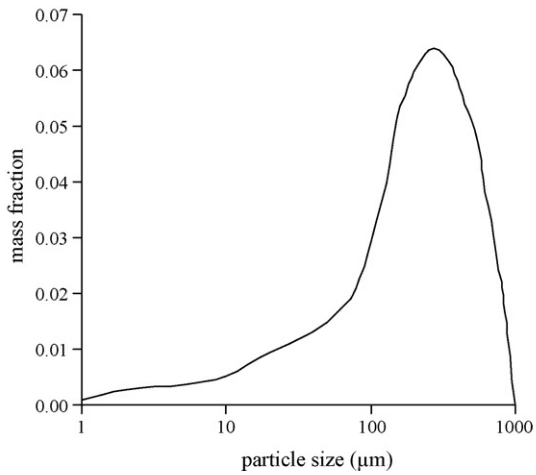

Fig. 2. Particle size distribution in the influent of the standard sedimentation tank.

not taken into account by the optical model. The latter is checked by means of the obscuration level, which should be inside a certain level. The possibility of misinterpreting the size distribution of open porous floccules by assuming them as compact spheres is well known in the literature but at present laser diffraction techniques are acceptable since there are no better alternatives [19].

Fig. 2 presents the measured particle size distribution in the influent of the standard sedimentation tank. The figure represents average values of three measurements conducted for the three sedimentation tanks of the plant. The repeatability expressed as the average standard deviation of the three measurements was 1.3%.

2.4. 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 not only are porous but it is well known that they have quite irregular shapes with spatial 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, D, is a 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 to 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 [20] to purely empirical relations. According to Gmachowski [21], the ratio of the resistance experienced by a floc to that of an equivalent solid sphere (f_{eff}) can be expressed as follows:

$$f_{\mathrm{eff}} = \sqrt{1.56 - \left(1.728 - \frac{D}{2}\right)^2} - 0.228 \qquad (1)$$

It is found that aggregates generated in water treatment processes exhibit a fractal dimension ranging between 2.2 and 2.6 [21–22], so the resistance coefficient, f_{eff}, varies between 0.85 and 0.95. This estimation agrees also with the theoretical results for the drag coefficient of fractal aggregates given by Vanni [23] 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 f_{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 0.90 and the apparent density was then estimated as 1066 kg m^{-3} by requiring the final computed settling effectiveness to coincide with the measured settling effectiveness of 90%. This is a typical effective density value for the aggregates met in water treatment applications [9]. The flow chart of this computations sequence is presented in Fig. 3.

2.5. Simulation

To limit computational power requirements, the circular settling tank was modeled in 2D. The major assumption in the development of the model is that the flow field is the same for all angular 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. A grid dependency study was performed to eliminate errors due to the coarseness of the grid and also to determine the best compromise between simulation accuracy, numerical stability, convergence, and computational time. In addition, the mesh density was chosen such that the grid

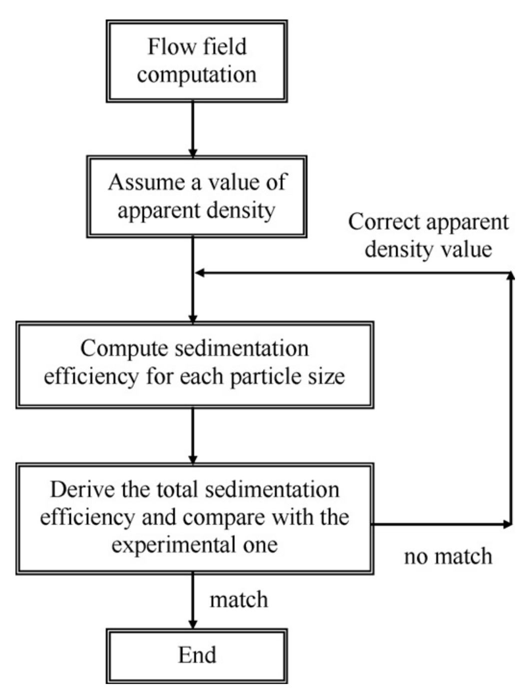

Fig. 3. Flow chart of the computations sequence.

was finest where velocity gradients are expected to be largest. The selected grid was comprised of 137,814 quadrilateral elements. Two other grids (one finer with 216,850 elements and one coarser with 11,170 elements) were also used to determine the effect of the overall grid resolution on predictions. While the predictions obtained using the coarse grid were found to be different from those resulting from the selected one, the difference between the predictions made by the selected and fine grids were insignificant. As a result, the solutions from the grid of 137,814 quadrilateral elements were considered to be grid independent.

The segregated solution algorithm was selected. The SST k–ω 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 [24]. The used discretisation schemes 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. Adams and Rodi [12] pointed out that for real settling tanks the walls can be considered as being smooth due the prevailing low velocities and the correspondingly large viscous layer. Consequently, the standard wall functions as proposed by Launder and Spalding [25] 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. Then the particles, whose density and size could be assigned at will, are released from the inlet and are tracked along their trajectories. The particles reaching the bottom were deemed trapped whereas the rest were considered escaped. Particle tracking is fast and 25,000 particles could be tracked in less than 10 min once the flow field had been computed.

The number of particles was selected after many trials in order to combine the solution accuracy with short computing time for convergence. Fig. 4 reveals that, as the number of particles increased from 2000 to 25,000, the number of iterations needed for the model to converge decreased, whereas an even higher number would not yield any significant improvement. Therefore, a number of 25,000 particles was selected as a suitable one. 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 particle number balance calculated at each iteration of the run. The convergence was achieved when the percentage of particles calculated to exit the tank no longer changed.

The settling tank was simulated for a specific set of conditions used in the Thessaloniki treatment plant 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 0.085 m s^{-1}, whereas the inlet turbulence intensity was set at 4.5%. The outlet was specified as a constant pressure outlet with a turbulence intensity of 6.0%. The water flow rate was 0.6 m^3 s^{-1}. Based on this rate, the inlet flow rate of particles was estimated as 0.15 kg s^{-1} using a measured solids concentration of 250 mg L^{-1}, whereas the primary particle density was 1066 kg m^{-3}.

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 (Fig. 2). The number of classes was selected in order to combine the solution accuracy with short computing time. Two other numbers, 6 and 16, were tested. While the predictions obtained using 6 classes of particles were found to be different from those resulting from the 13 classes, the difference between the predictions made by the 13 and the 16 classes were insignificant. Therefore, a number of 13 classes was selected as a suitable one. Within each class the particle diameter is assumed to be constant (Table 1). As it 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, n, was based on the calculation of the percent of solids

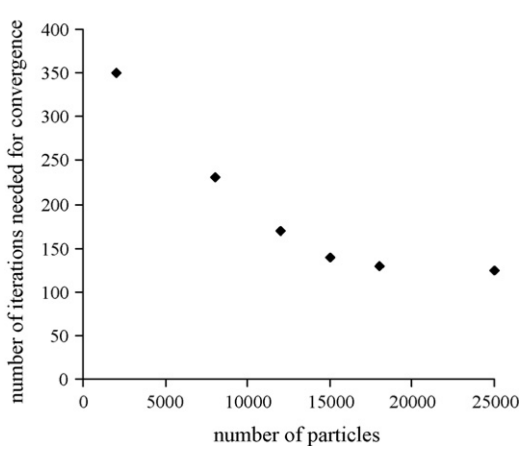

Fig. 4. Effect of particles number on the number of iterations required to achieve a converged solution.

Table 1
Classes of particles used to account for the total suspended solids in the sedimentation tank

Class	Range of particle size (μm)	Mean particle size (μm)	Mass fraction
1	10–30	20	0.025
2	30–70	50	0.027
3	70–90	80	0.039
4	90–150	120	0.066
5	150–190	170	0.095
6	190–210	200	0.115
7	210–290	250	0.126
8	290–410	350	0.124
9	410–490	450	0.113
10	490–610	550	0.101
11	610–690	650	0.077
12	690–810	750	0.057
13	810–890	850	0.040

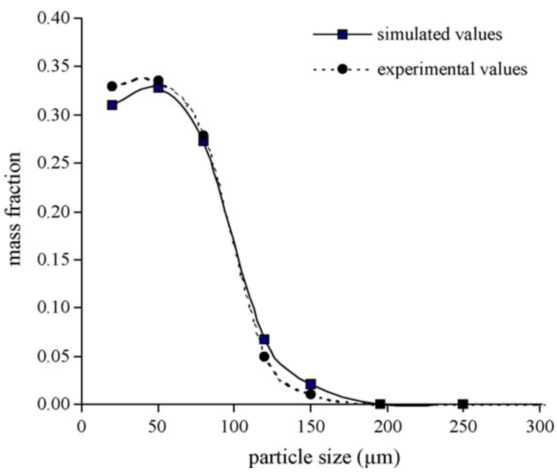

Fig. 5. Simulated and experimental particle size distribution in the effluent of the standard sedimentation tank.

settled for each particle size class, n_i:

$$n = \frac{\sum_{i=1}^{n}(c_i n_i)}{\sum_{i=1}^{n} c_i} \quad (2)$$

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 feed.

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 n_i calculated from the 13 simulations, the sedimentation efficiency could be calculated for any different particle size distribution in the influent.

3. Results and discussion

3.1. Model validity

As far as the CFD model validity is concerned, Fig. 5 presents a comparison between the experimentally measured and the simulated values of the floc size distribution in the effluent of the standard tank. Apparently, there is a good agreement between measured and predicted values.

3.2. Flow pattern

The removal efficiency in settling tanks depends on the physical characteristics of the suspended solids as well as on the flow field and the mixing regime in the tank. Therefore the determination of flow and mixing characteristics is essential for the prediction of the tank efficiency. Fig. 6 presents the predicted streamlines for the standard and the modified tank. The displayed simulations made with solids present refer to particles of class size 4 (Table 1). The influent, after impinging on the standard flow control baffle at point A, is deflected downwards to the tank bottom. The flow splits at point B on the bottom of the tank, producing a recirculation eddy at C. Generally, the flow pattern is characterized by a large recirculation region spanning a large part of the tank from top to bottom. Three smaller recirculation regions are also found; two at the top of the tank near the entry and exit points of the liquid stream and one at the bottom right-hand side of the tank just above the cavity where the sludge gathers before leaving the tank. These regions have

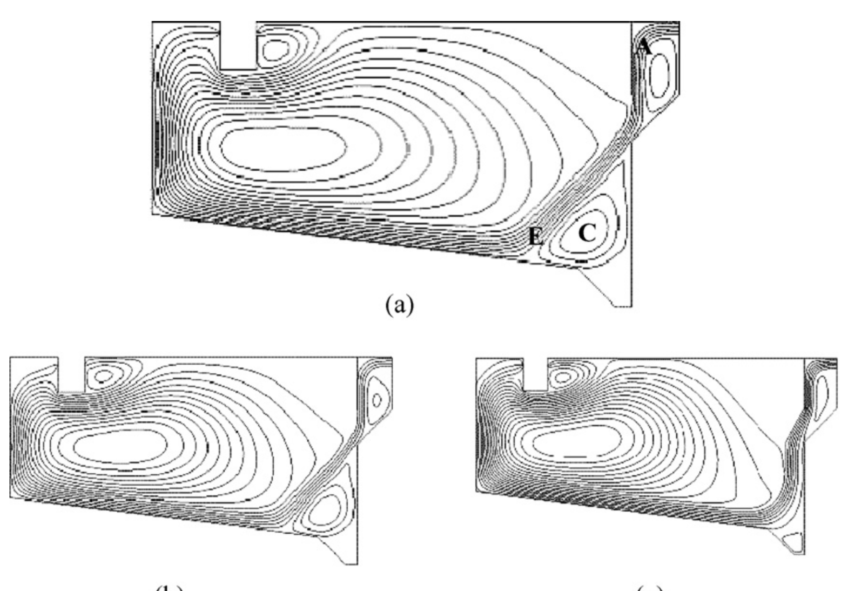

Fig. 6. Calculated streamlines for the standard and the modified sedimentation tank. (a) Standard tank, water, (b) standard tank, water + particle class size 4 and (c) modified tank, water + particle class size 4.

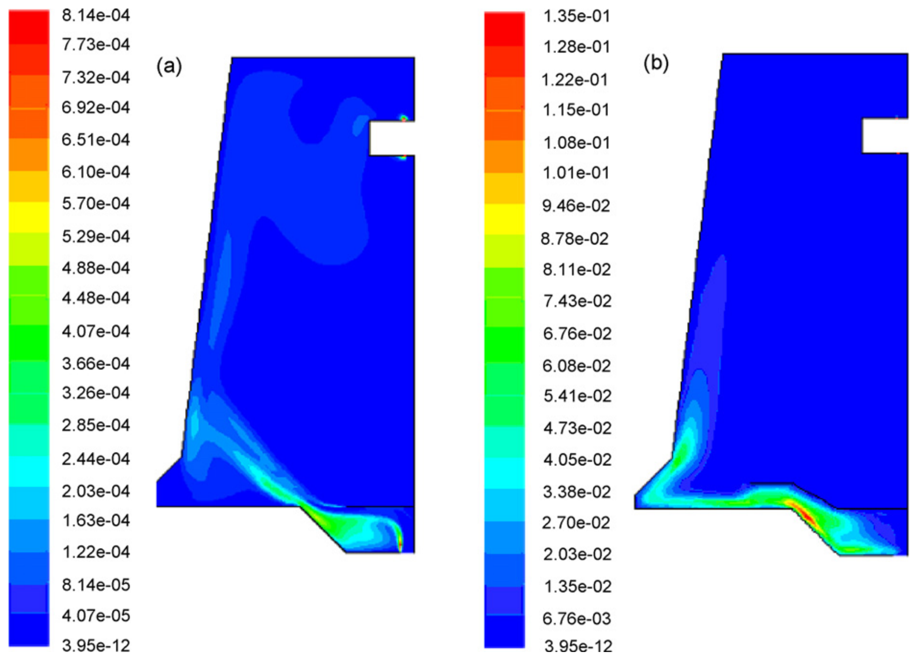

Fig. 7. Contours of turbulent kinetic energy ($m^2 s^{-2}$) for the standard (a) and the modified (b) sedimentation tank for particle class size 1.

a substantial impact on the hydrodynamics and the efficiency of the sedimentation tank. The same behavior was observed by Stamou [26] in his flow velocity predictions in a settling tank using a curvature-modified k–ε model.

The above-mentioned observations are in agreement with findings of Zhou and McCorquodale [27], who studied numerically the velocity and solids distribution in a clarifier. According to another numerical work [9], in secondary clarifiers there is a circular current showing: (1) forward flow velocities in the zone close to the tank bottom, (2) backward flow velocities in the upper zone of the tank, (3) higher forward flow velocities in the inlet than in the rim region, (4) higher backward flow velocities in the inlet than in the outlet region, (5) vertical currents downwards to the tank bottom in the inlet region, and (6) vertical currents upwards to the water level in the outlet region. For the case of secondary clarifiers with high-suspended solids concentration, a density current exists due to a higher density of the incoming suspension. This current sinks toward the sludge blanket right after leaving the inlet structure and flows towards the tank rim. As a result, backward velocities

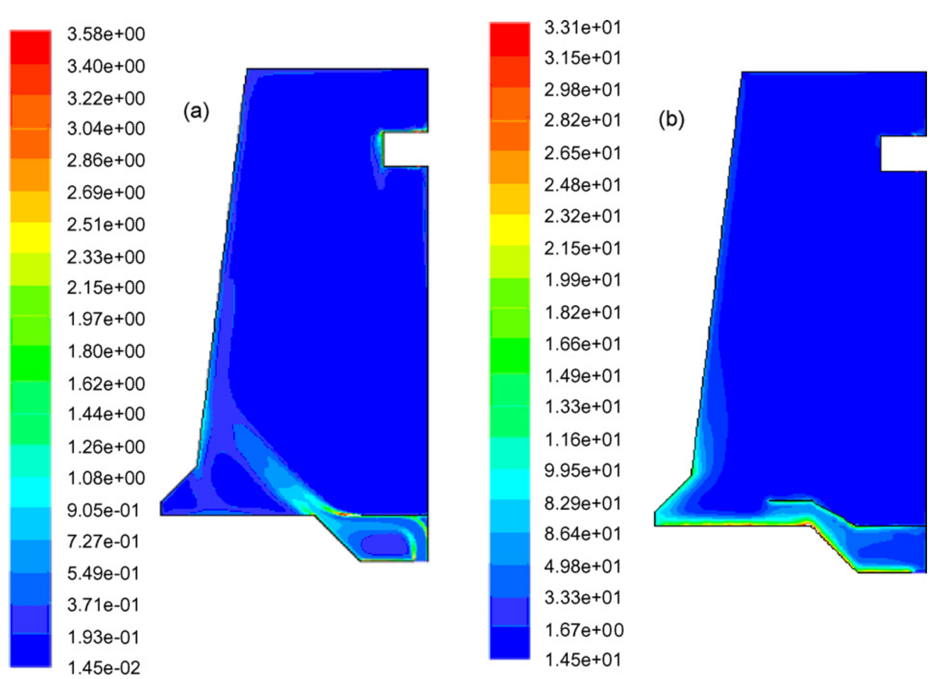

Fig. 8. Contours of specific dissipation rate (s^{-1}) for the standard (a) and the modified (b) sedimentation tank for particle class size 1.

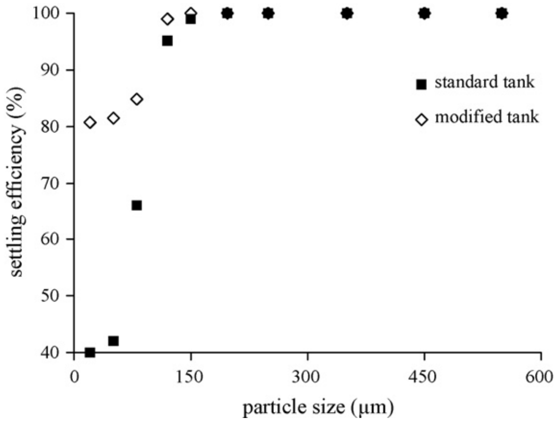

Fig. 9. Predicted percents of solids settled for each particle size class.

are induced in the upper water zone following the continuity equation.

A number of researchers have observed the solids-cascading phenomenon in the clarification of concentrated activated sludge in either theoretical simulations or experimental works. However, a density current is not observed in this work. As it can be concluded comparing Figs. 6(a) and (b), the particles do not affect the flow field. This observation is attributed to the small particle loading in our sedimentation tank and is similar to that made by Kim et al. [2], who worked in a secondary clarifier with a neutral density influent flow. On the contrary, in the case of high inlet fluid density (high solids concentration) combined with a low fluid velocity, the horizontal inlet flow does not even reach the flow control baffle, but plunges down toward the tank bottom as a density waterfall due to the low Froude number.

In the modified tank, the flow split point B moves more to the right of the tank bottom compared to the position in the standard tank and the recirculation zone above the sludge corner is now very small. It appears that the extended baffle does not affect the flow pattern or the particle trajectories throughout the tank or at the exit. Neither does it affect the particle settling patterns on the bottom of the tank. The difference is mostly restricted at the entrance section and near the bottom rim of the tank, so that the upward flow in the downstream zone is only slightly different.

In general, the extended baffle appears to provide better influent mixing and isolation between the tank influent and effluent than that in the original tank design, thus significantly enhancing sedimentation. In addition, it allows a better utilization of the full tank depth than in the standard design that leads to better separation between the influent and effluent along the vertical direction. Studies by Zhou and McCorquodale [27] revealed the importance of a baffle in dissipating the kinetic energy of the incoming flow and reducing short-circuiting and indicated that the location of the baffle has a pronounced effect on the nature of the flow. According to them, the lower the kinetic energy dissipation rate, the more intense is the recirculation zone. The effect of the extended baffle on turbulent kinetic energy and specific dissipation rate is presented in Figs. 7 and 8. As it can be seen, the extended baffle increases the kinetic energy and the dissipation rate in the inlet baffling region and, as a consequence, weakens the currents in this region.

3.3. Solids distribution

It is essential to present first the computed variation in settling efficiency with respect to particle size. Fig. 9 presents the predicted percents of solids settled for different tested particle size classes. As it can be inferred, the theoretical settling efficiency tends to non-zero (in fact, relatively large) values as the particles 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 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 shake of completeness of the simulation

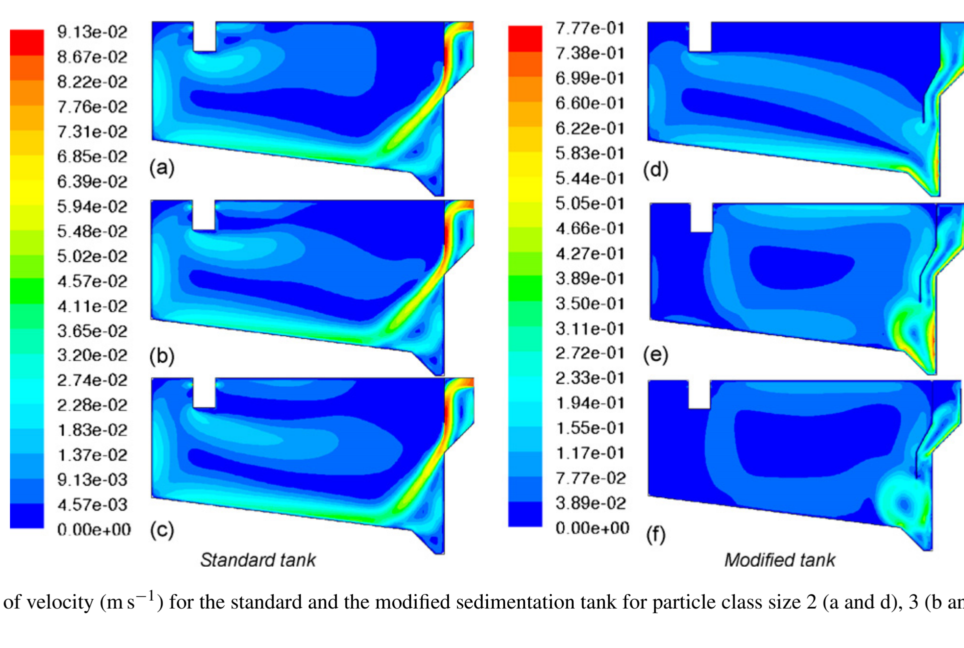

Fig. 10. Contours of velocity (m s^{-1}) for the standard and the modified sedimentation tank for particle class size 2 (a and d), 3 (b and e) and 4 (c and f).

procedure. The easiest way to accommodate the realistic behavior of a decreasing settling efficiency as particles 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 particles size decreases the effect of gravity decreases leading the probability from unity to the inverse of the stability ratio well known to the studies of small particle deposition from turbulent flows [28]. Nevertheless, despite the aforementioned improvement, the present CFD model provides a good overall description of the system behavior.

The percents presented in Fig. 9 result in an overall settling efficiency of 90.4 and 98.6% for the standard and the modified tank, respectively. As it can be seen, the model predicts highly distinct concentration for different classes of particle; lower removal rate for the smallest and higher removal rate for the heaviest particles. The percents for classes 6–13 (see Table 1) are very close to 100% indicating that the particles with the eight highest settling velocities would be settled almost completely regardless of the configuration used. 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). This observation is similar to that obtained by Huggins et al. [1], who used a CFD model to evaluate the impact of potential raceway design modifications on the in-raceway settling of solids.

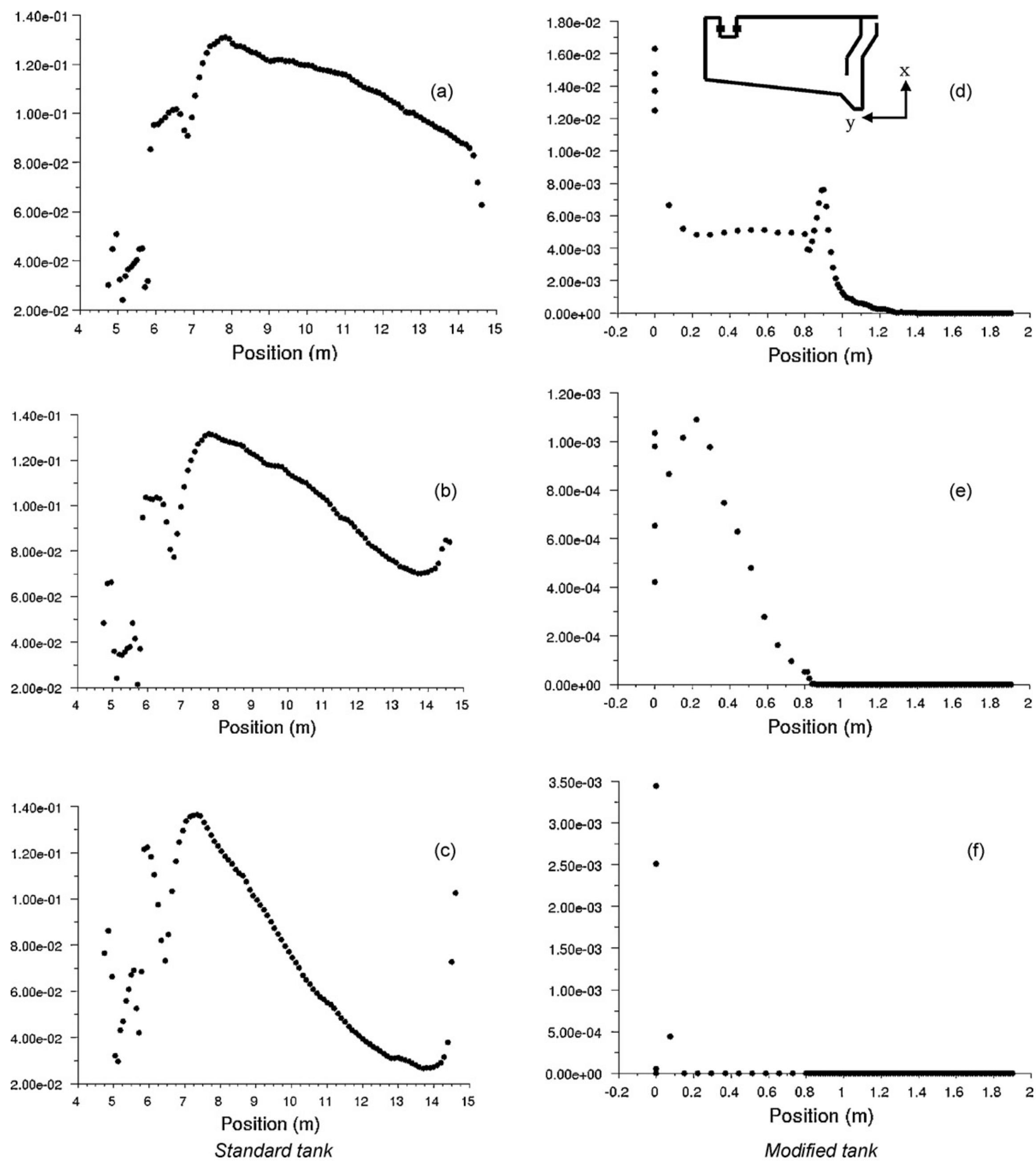

Fig. 11. Flocs concentration (kg m^{-3}) along the tank bottom for the standard and the modified tank for particle class size 2 (a and d), 3 (b and e) and 4 (c and f).

Although the increase in the overall effectiveness seems small it corresponds to an estimated reduction in the solids exiting the tank of approximately 520 kg day^{-1} or to a reduction of about 85% of the solids in classes 1–13 that exit the tank. These values are greater than those reported by other researchers. Huggins et al. [1], who tested a number of potential raceway design modifications noticed that by adding a baffle 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 [29] used an additional baffle at mid-radius extending from the floor upwards to mid-depth and observed a reduction of 38% in effluent concentration.

According to Huggins et al. [1], the particle settling velocity has a significant impact on the settling efficiency for a given raceway design. These authors argued that an important consideration in trying to improve the settling of particles is to reduce the mass fraction of solids with settling velocities below 0.01 m s^{-1}. However, since influent solids load is usually uncontrollable one should focus instead on the design of a proper baffle, which will improve solids settling by forcing them to reach fast the bottom of the tank. Fig. 10 shows contours of velocity for the standard and the modified tank. It must be noted that in this and following figures only results of simulations with small particles (less than 150 μm) are presented since for larger particles settling is satisfactory even with the standard tank. Indeed, in Fig. 10 one can see that particle settling velocity increases for such small particles when using the extended baffle.

The effect of an extended baffle is also displayed in Figs. 11 and 12 that show flocs concentration along the tank

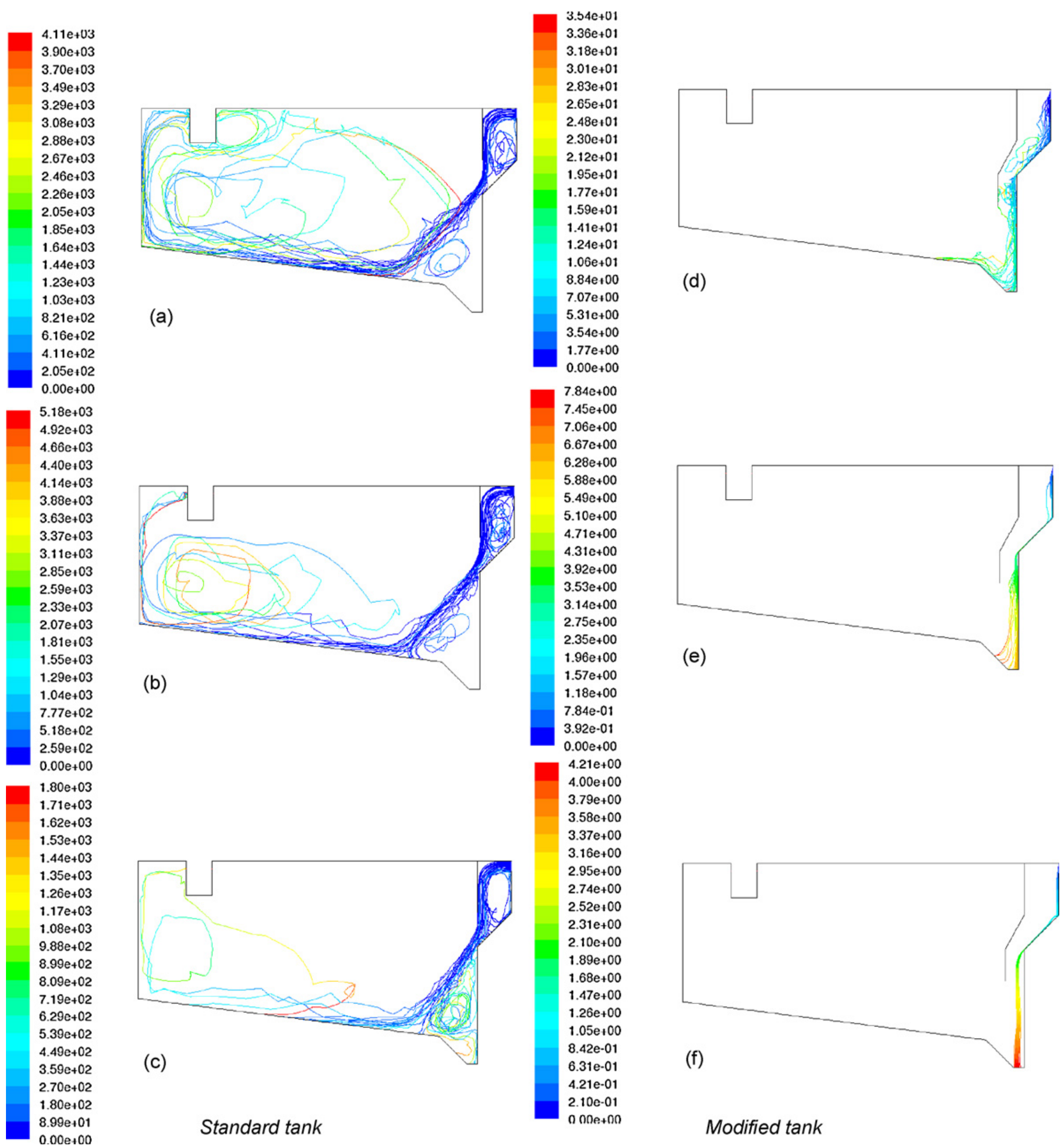

Fig. 12. Flocs pathlines colored by residence time for the standard and the modified tank for particle class size 2 (a and d), 3 (b and e) and 4 (c and f).

bottom and pathlines colored by particle residence time, respectively, for particle classes 2, 3 and 4. In Fig. 11 the zero position of the horizontal axis is set at the right-hand end of the tank bottom. Clearly, the modified tank allows flocs to settle at much short distances from the right-hand corner of the tank. This result is also seen in Fig. 12 where one can further notice that flocs that have failed to settle may be dragged towards the exit. This diminishes the overall settling efficiency of the tank. On the whole, the simulation results demonstrate quantitatively the drastic effect of particle velocity on sedimentation effectiveness. Higher settling velocities lead to more effective sedimentation. However, even small differences in particle settling velocity can cause large changes in the percent of settled particles.

4. Conclusions

This work deals with the development 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 field and different particle classes) than the corresponding tools employed to simulate primary and secondary wastewater settling tanks. The present code is modified based on data from a real sedimentation tank. Then it is used to assess the significance of extending the feed flow control baffle of this particular sedimentation tank.

The results show that an extended baffle forces the solids to move faster towards the bottom of the tank and decreases the inlet recirculation zone, thus yielding significantly enhanced sedimentation. Although the increase in the overall effectiveness by this baffle may show only a small change, this actually reflects a reduction of the effluent solids of estimated around 85%. In general, CFD can be a powerful tool for troubleshooting problems, particularly those associated with flow patterns in a sedimentation tank. The results of this work give an insight, which can be used to investigate novel designs or different operating conditions, such as temperature variation, for production-scale tanks. Of course, because of the complexity of the processes taking place, CFD will not completely replace experimental testing and the partly empirical nature of the design process. Traditional techniques will continue to be used for routine design, but CFD is invaluable for backing up this work and for investigating novel tank designs.

Acknowledgements

Thanks are due to the Konstandinidis S.A. technical company (constructor and operator of plant) scientific personnel: A. Papaioannou (Chem. Eng.), A. Samaras (Mech. Eng.) and M. Papagiannis (Chem.) and E. Samara (Computer Technician) for their help in many ways.

References

[1] D.L. Huggins, R.H. Piedrahita, T. Rumsey, Use of computational fluid dynamics (CFD) for aquaculture raceway design to increase settling effectiveness, Aquacult. Eng. 33 (2005) 167–180.
[2] H.S. Kim, M.S. Shin, D.S. Jang, S.H. Jung, J.H. Jin, Study of flow characteristics in a secondary clarifier by numerical simulation and radioisotope tracer technique, Appl. Radiat. Isotopes 63 (2005) 519–526.
[3] P. Larsen, On the hydraulics of rectangular settling basins, Report No. 1001, Department of Water Research Engineering, Lund Institute of Technology, Lund, Sweden, 1977.
[4] D.R. Shamber, B.E. Larock, Numerical analysis of flow in sedimentation basins, J. Hydr. Div. ASCE 107 (HY5) (1981) 575–591.
[5] J.A. McCorquodale, E.M. Yuen, Z. Vitasovic, R. Samstag, Numerical simulation of unsteady conditions in clarifiers, Water Poll. Res. J. Can. 26 (1991) 201–222.
[6] J.A. McCorquodale, S. Zhou, Effects of hydraulic and solids loading on clarifier performance, J. Hydr. Res. 31 (1993) 461–477.
[7] S. Zhou, J.A. McCorquodale, A.M. Godo, Short circuiting and density interface in primary clarifiers, J. Hydr. Eng. 120 (1994) 1060–1080.
[8] P. Krebs, D. Vischer, W. Gujer, Inlet-structure design for final clarifiers, J. Environ. Eng. 121 (1995) 558–564.
[9] A. Deininger, E. Holthausen, P.A. Wilderer, Velocity and solids distribution in circular secondary clarifiers: full scale measurements and numerical modelling, Water Res. 32 (1998) 2951–2958.
[10] E. Imam, J.A. McCorquodale, J.K. Bewtra, Numerical modeling of sedimentation tanks, J. Hydr. Eng. 109 (1983) 1740–1754.
[11] A.I. Stamou, E.A. Adams, W. Rodi, Numerical modelling of flow and settling in primary rectangular clarifiers, J. Hydr. Res. 27 (1989) 665–682.
[12] E.W. Adams, W. Rodi, Modelling flow and mixing in sedimentation tanks, J. Hydr. Eng. 116 (1990) 895–913.
[13] D.A. Lyn, A. Stamou, W. Rodi, Density currents and shear induced flocculation in sedimentation tanks, J. Hydr. Eng. 118 (1992) 849–867.
[14] P.H. Frey, Hydrodynamics fields and solid particle transport in a settling tank, J. Hydr. Res. 31 (1993) 763–776.
[15] J.J. Van der Walt, Is a sedimentation tank really that simple? in: Proceedings of the WISA Biennale Conference and Exhibition, Cape Town, South Africa, 1998.
[16] B. de Clercq, P.A. Vanrolleghem, Computational fluid dynamics in wastewater treatment Med. Fac. Landb. Toegep. Biol. Wet. 67 (2002) 15–19.
[17] S. Berres, R. Burger, E.M. Tory, Applications of polydisperse sedimentation models, Chem. Eng. J. 111 (2005) 105–117.
[18] D. Wilkinson, B. Waldie, M.I. Mohamad Nor, H.Y. Lee, Baffle plate configurations to enhance separation in horizontal primary separators, Chem. Eng. J. 77 (2000) 221–226.
[19] I. Nopens, T. Koegst, K. Mahieu, P.A. Vanrolleghem, PBM and activated sludge flocculation: from experimental data to calibrated model, Environ. Ener. Eng. 51 (2005) 1548–1557.
[20] C.P. Chu, D.J. Lee, J.H. Tay, Floc model and intrafloc flow, Chem. Eng. Sci. 60 (2005) 565–575.
[21] L. Gmachowski, Aggregate structure and hydrodynamics of aggregated systems, Colloids Surf. A: Physicochem. Eng. Aspects 255 (2005) 105–110.
[22] D.J. Lee, G.W. Chen, Y.C. Liao, C.C. Hsieh, On the free-settling test for estimating activated sludge floc density, Water Res. 30 (1996) 541–550.
[23] M. Vanni, Creeping flow over spherical permeable aggregates, Chem. Eng. Sci. 55 (2000) 685–698.
[24] D.C. Wilcox, Turbulence Modeling for CFD, DCW Industries, Inc., La Canada, California, 1998.
[25] B.E. Launder, D.B. Spalding, The numerical computation of turbulent flows, Comp. Meth. Appl. Mech. Eng. 3 (1974) 269–289.
[26] A.I. Stamou, On the prediction of flow and mixing in settling tanks using a curvature-modified k–ε model, Appl. Math. Model. 15 (1991) 351–358.

[27] S. Zhou, J.A. McCorquodale, Influence of skirt radius on performance of circular clarifiers with density stratification, Int. J. Numer. Meth. Fluids 14 (1992) 919.
[28] M. Kostoglou, A.J. Karabelas, Comprehensive modeling of precipitation and fouling in turbulent pipe flow, Indus. Eng. Chem. Res. 37 (1998) 1536–1550.
[29] R.M. Crosby, Hydraulic characteristics of activated sludge-secondary clarifiers, EPA 600/2-84-131, NTIS No. PB-84-229665, U.S. EPA, Municipal and Environmental Research Laboratories, Cincinnatti, OH, 1984.