Desalination and Water Treatment
www.deswater.com
doi: 10.1080/19443994.2016.1157521

57 (2016) 25950–25959
November

Taylor & Francis
Taylor & Francis Group

Simulation of effect of a baffle on the flow patterns and hydraulic efficiency in a sedimentation tank

Yuling Liu, Pei Zhang, Wenli Wei*

State Key Laboratory base of Eco-Hydraulic Engineering in Arid Area (SKLEHE), Xi'an University of Technology, Xi'an, Shaanxi 710048, China, Tel. +86 18591999882; email: liuyuling@xaut.edu.cn (Y. Liu), Tel. +86 13991987708; email: Peizhang@126.com (P. Zhang), Tel. +86 15596886263; email: wei_wenli@126.com (W. Wei)

Received 20 April 2015; Accepted 18 February 2016

ABSTRACT

This paper is concerned with a solid–liquid two-phase model combining with the k–ε turbulent model for the numerical simulation of the effect of a baffle on the hydraulic characteristics and solid removing rate of a sedimentation tank. The computed flow patterns, the distributions of solid concentration along vertical lines on the cross sections were obtained in the sedimentation tank by numerical simulation. Comparisons of the computed results show that: (1) with the horizontal baffle-distance between 0.5 and 2.5 m, the removal rate increases with a decrease in the horizontal baffle-distance, and ranges from 82.63 to 92.6%. Therefore, the horizontal baffle-distance between 0.5 and 2 m is recommended, (2) with an increase of baffle submerged-depth from 0.5 to 2 m, the removal rate at outlet increases, and especially, with the baffle-depth from 0.5 to 1 m, the removal rate increases obviously; however, with the baffle submerged-depth from 2 to 2.5 m, the removal rate decreases slightly, which shows that the baffle does not need a much greater submerged-depth, and 2 m is recommended. The research results can provide a useful reference for the design of sedimentation tanks.

Keywords: Sedimentation tank; Baffle; Simulation; Hydraulic characteristics; Solid removing rate

1. Introduction

The baffle is an important part of a sedimentation tank, which decreases the inlet recirculation zone and enhances the settling of solids by directing them toward the bottom of the tank with high velocities. Installing a baffle in a sedimentation basin and its effects on the basin hydraulic efficiency are challenging subjects on which many studies have been conducted, and various results have been obtained about the baffle effectiveness in sedimentation tanks [1,2]. Sarkar et al. [3] investigated the performance of the inclined plate settlers for aquaculture waste. Razmi et al. [4] studied the effects of the baffle position on the performance of a primary settling tank experimentally and numerically. Fan et al. [5] modeled the solid–liquid two-phase turbulent flow in a tank with a three-dimensional two-fluid model. Using computational fluid dynamics (CFD), the velocity profile and distribution of solid concentration are obtained. The use of different baffles in the same tank was also simulated. With the inclusion of the baffle, the

*Corresponding author.

distribution of the solid concentration differs greatly. The significant effects of the baffle height and baffle location on the distribution of solid concentrations are also discussed. Ramin et al. [6] developed a new settling velocity model, including hindered, transient and compression settling for secondary settling tanks (SSTs). A 2D axisymmetric CFD model of a circular SST containing the new settling velocity and rheological model was validated with full-scale measurements. Finally, it was shown that the representation of compression settling in the CFD model can significantly influence the prediction of sludge distribution in the SSTs under dry- and wet-weather flow conditions. Doroodchi et al. [7] investigated the effect of inclined plates on the expansion behavior of solids suspensions in liquid fluidized beds. Amini et al. [8] use a three-dimensional numerical model to simulate the velocity fields in a scaled-down physical model, which finds that the CFD model could accurately predict the hydrodynamics of the tank. Thus, further investigations are conducted in another tanks with different number of baffles to explore the effect of their numbers on the hydraulic characteristics of the unit using the developed CFD model, which shows the existence of recirculation areas behind the baffles in a way that the extension of these regions dwindles on increasing the number of baffles, leading the fluid flow to approach plug-flow conditions. The removal efficiency in settling tanks depends on the physical characteristics of the suspended solids (SSs), the flow field, and the mixing regime in the tanks. Therefore, the determination of flow characteristics is essential for the prediction of the tank efficiency. The baffle has a big effect on the hydraulic characteristics in a sedimentation tank; however, few studies deal with this problem. Therefore, the paper uses a solid–liquid two-phase mixture model along with the standard k–ε turbulence model in the CFD package FLUENT 6.2.16 to investigate the effect of a baffle on the hydraulic characteristics and solid removing rate of a sedimentation tank, which can provide a reference in designing sedimentation tanks.

2. Mathematical model

2.1. Governing equations

Two-fluid flow models can be used to simulate the two-phase flow in settling tanks. The mixture model is a simplified two-fluid flow model, which uses the local equilibrium of a small spatial scale for solving the mixed phase momentum, continuity and energy equations, the second-phase volume fraction, slip velocity, and drift velocity. Considering that the model is a simple one with a small amount of calculation and reliable simulation results [9], we used the mixture model.

The time-averaged continuity equation for solid–liquid two-phase flows is expressed by [9,10]:

$$\frac{\partial}{\partial t}(\rho_m) + \nabla \cdot (\rho_m \vec{v}_m) = 0 \tag{1}$$

$$\vec{v}_m = \frac{\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_k}{\rho_m} \tag{2}$$

where ρ_k is the k phase density; $\rho_m = \sum_{k=1}^{n} \alpha_k \rho_k$, is the mixed phase density; $\vec{v}_m$ is the mixed phase mass-averaged velocity; $\vec{v}_k$ is the k phase mass-average velocity; α_k is the k phase volume fraction.

The momentum conservation equation for solid–liquid two-phase flows is given by [9,10]:

$$\begin{aligned}\frac{\partial}{\partial t}(\rho_m \vec{v}_m) &+ \nabla \cdot (\rho_m \vec{v}_m \vec{v}_m) \\ &= -\nabla p + \nabla \cdot [\mu_{t,m}(\nabla \vec{v}_m + \nabla \vec{v}_m^{\mathrm{T}})] + \rho_m \vec{g} + \vec{F} \\ &\quad + \nabla \cdot \left(\sum_{k=1}^{n} \alpha_k \rho_k \vec{v}_{dr,k} \vec{v}_{dr,k} \right)\end{aligned} \tag{3}$$

where n is the number of phases; $\vec{F}$ is the volume force; $\vec{g}$ is the acceleration of gravity; $\mu_{t,m}$ is the mixed viscosity; $\vec{v}_{dr,k}$ is the k phase drifting velocity, and is defined as $\vec{v}_{dr,k} = \vec{v}_k - \vec{v}_m$; The slipping velocity, $\vec{v}_{pq}$, is defined as the p phase velocity relative to the q phase velocity, and $\vec{v}_{pq} = \vec{v}_p - \vec{v}_q$.

The relationship between the slipping velocity and the drifting velocity is:

$$\vec{v}_{dr,p} = \vec{v}_{pq} - \sum_{k=1}^{n} \frac{\alpha_k \rho_k}{\rho_m} \vec{v}_{qk} \tag{4}$$

The equation describing p phase volume fraction is written as [9]:

$$\frac{\partial}{\partial t}(\alpha_p \rho_p) + \nabla \cdot (\alpha_p \rho_p \vec{v}_m) = -\nabla \cdot (\alpha_p \rho_p \vec{v}_{dr,p}) \tag{5}$$

The standard k– turbulent model is written as [9,10]:

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla \cdot (\rho_m \vec{v}_m k) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \varepsilon \tag{6}$$

$$\frac{\partial}{\partial t}(\rho_m \varepsilon) + \nabla \cdot (\rho_m \vec{v}_m \varepsilon) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_\varepsilon} \nabla \varepsilon\right) + \frac{\varepsilon}{k}\left(C_{1\varepsilon} G_{k,m} - C_{2\varepsilon} \rho_m \varepsilon\right) \quad (7)$$

$$\mu_{t,m} = \rho_m C_\mu \frac{k^2}{\varepsilon} \quad (8)$$

$$G_{k,m} = \mu_{t,m}\left[\nabla \vec{v}_m + (\nabla \vec{v}_m)^{\mathrm{T}}\right] \cdot \nabla \vec{v}_m \quad (9)$$

where k is the turbulent kinetic energy and is the kinetic energy dissipation rate; C_μ, μ, μ, $C_{1,}$ and C_2 are empirical constants and have the values of 0.99, 1.0, 1.3, 1.44, and 1.92, respectively.

2.2. Computational physical model

Fig. 1 shows a schematic diagram of a horizontal rectangular sedimentation tank with a length of 25.5 m, whose entrance is 0.5 m high, and outlet is 0.3 m high. The mud bucket is 3 m deep, with an entrance of width 4 m, and with a bottom slope of 0.02 [10]. L_{b} is a horizontal distance from the water entrance, and h_{b} is the baffle submerged-depth; the effect of different horizontal distances and baffle submerged-depths on the flow patterns and hydraulic efficiency in the sedimentation tank is studied. The computational region is shown as Fig. 1(a), and the detail size of the tank is shown as Fig. 1(b), respectively.

2.3. Grid generation and grid-independent solution

The unstructured computational grid, as shown in Fig. 2, with 13,040 cells, was generated by the GAMBIT procedures. In order to guarantee a grid-independent solution, a relatively exhaustive grid-independence study has been performed with 3 grid configurations, the number of cells of which are 10,035, 13,040, and 14,509, respectively, which ensures that the number of cells, 13,040, represents a grid-independent solution.

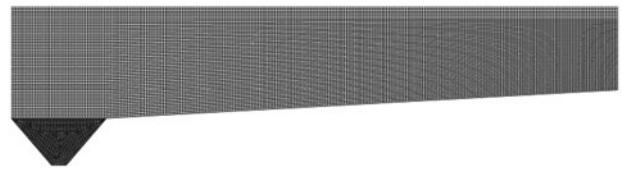


Fig. 2. Grid of the computational region.

2.4. Boundary and initial conditions and solution method of equations

For the sedimentation tank (in Fig. 1(b)), the open boundary conditions are applied at the influent inlet and effluent outlet. At the inlet section, the velocity is specified according to the inflow rate, and the inlet sludge concentration is also specified; at the outlet section, the boundary pressure is atmospheric, and the normal gradient of the sludge concentration is zero. No-slip boundary conditions are assigned for all the walls, including the bottom surface and the side wall. A "rigid-lid" assumption and slip wall boundary condition are applied to the surface of mixture. The clear water in rest filled in the sedimentation tank is taken as the initial condition.

The control equations are discretized by the finite volume method, and solved with the algorithm of pressure-implicit with splitting of operators (PISO).

3. Results and discussion

3.1. Analysis of the position effect of the baffle on the flow fields and the concentration field distributions

Besides the submerged depth of a baffle, its position is an important factor affecting the efficiency of sedimentation tanks. The hydraulic characteristics of the sedimentation tank was studied with the baffle submerged to a depth of 1.5 m, and at a distance of 0.5, 1, 1.5, 2, and 2.5 m away from the water inlet, respectively. Fig. 3(a) shows that when the horizontal position of the baffle is 0.5 m away from the water inlet, no recirculation zone appears upstream of the

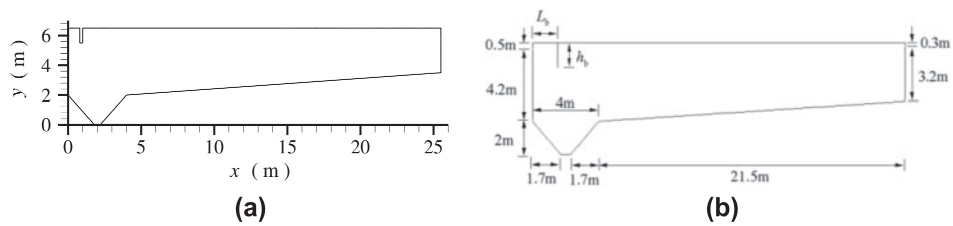


Fig. 1. A schematic diagram of the tank: (a) computational region of the tank and (b) detail size of the tank.

baffle, but only a small one appears right below the baffle. This phenomenon results from the baffle being much closer to the water inlet. Near the bottom of the tank, the flow pattern is complicated owing to a lot of vortexes of different sizes. When the horizontal position of the baffle is increased to 1 m, as shown in Fig. 3(b), a small recirculation zone appears upstream of the baffle, and the size of the recirculation zone downstream of the baffle remains unchanged, but its position goes up. And a smaller recirculation zone appears below the water outlet. When the horizontal position of the baffle is increased to 1.5 m, as shown in Fig. 3(c), two recirculation zones appear upstream of the baffle, and the recirculation zone below the water outlet also appears. When the horizontal position of the baffle is increased to 2 m, as shown in Fig. 3(d), the recirculation zone disappears completely downstream of the baffle, but upstream of the baffle only one recirculation zone exists, and the recirculation zone below the water outlet becomes larger. When the horizontal position of the baffle is increased to 2.5 m, as shown in Fig. 3(e), the recirculation zone upstream of the baffle becomes larger than any before, and the recirculation zone below the water outlet enlarges and moves upward slightly.

In order to better describe the effect of the horizontal position of the baffle on the concentration field distributions, the c*oncentration distributions* along vertical lines on different sections ($x = 0.2$, 4, 10, 15, 20, and 24 m) are analyzed quantitatively, as shown in Fig. 4.

Fig. 4 shows that the clear water zone near the water surface of the sedimentation tank reduces when the baffle-horizontal distance away from the water inlet increases. When the horizontal positions of the baffle are 0.5, 1, 1.5 m away from the water inlet, respectively, nearly no sludge appears in the region near the water surface, but when the horizontal positions are 2 and 2.5 m, some sludge appears. That is to say that the horizontal positions of 0.5, 1, and 1.5 m are better than that of 2 and 2.5 m.

SS values at clear water outlet and the corresponding removal rates of solids were computed and are shown in Table 1 with different horizontal baffle-positions of 0.5, 1, 1.5, 2, and 2.5 m away from the water inlet.

Table 1 shows that when the horizontal baffle-distance is between 0.5 and 2.5 m, the removal rate increases with a decrease in the horizontal baffle-distance away from the water inlet. The corresponding removal rates of solids with different horizontal baffle-positions of 0.5, 1, 1.5, 2, and 2.5 m away from the water inlet are 92.65, 92.69, 89.32, 82.63, and 79.41%, respectively.

When the horizontal distance of the baffle away from the water inlet increases from 0.5 to 1 m, the removal rate is nearly unchanged and is about 92.6%; but when it increases from 1 to 1.5 m, the removal rate goes down from 92.6 to 89.32%, and the net change is about 3.3%; and also, with the horizontal baffle-position being from 1.5 to 2 m, the net change is 6.69%, and from 2 to 2.5 m, it is 3.22%.

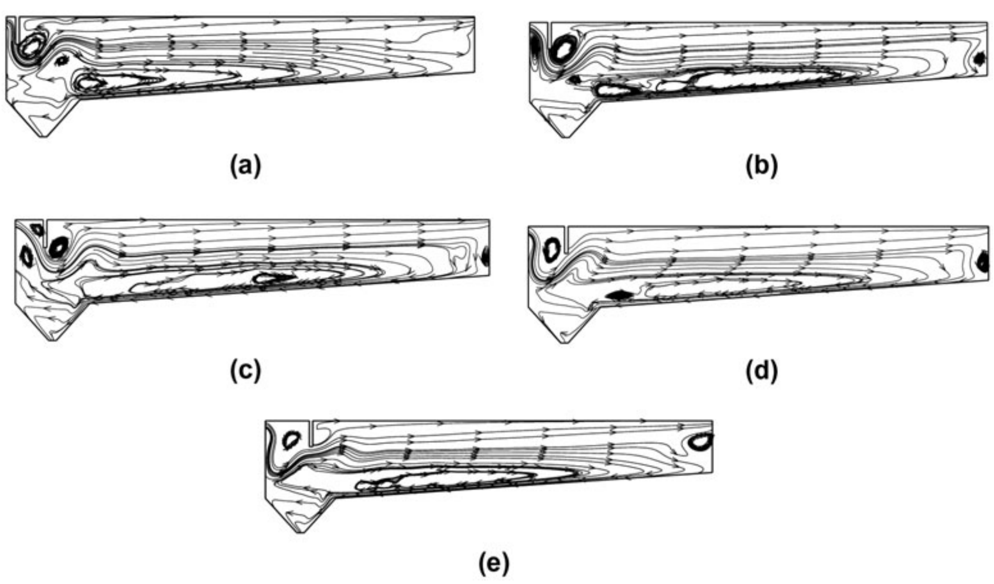


Fig. 3. Computed streamlines of horizontal baffle-distances of (a) 0.5 m, (b) 1 m, (c) 1.5 m, (d) 2 m, and (e) 2.5 m away from the water inlet.

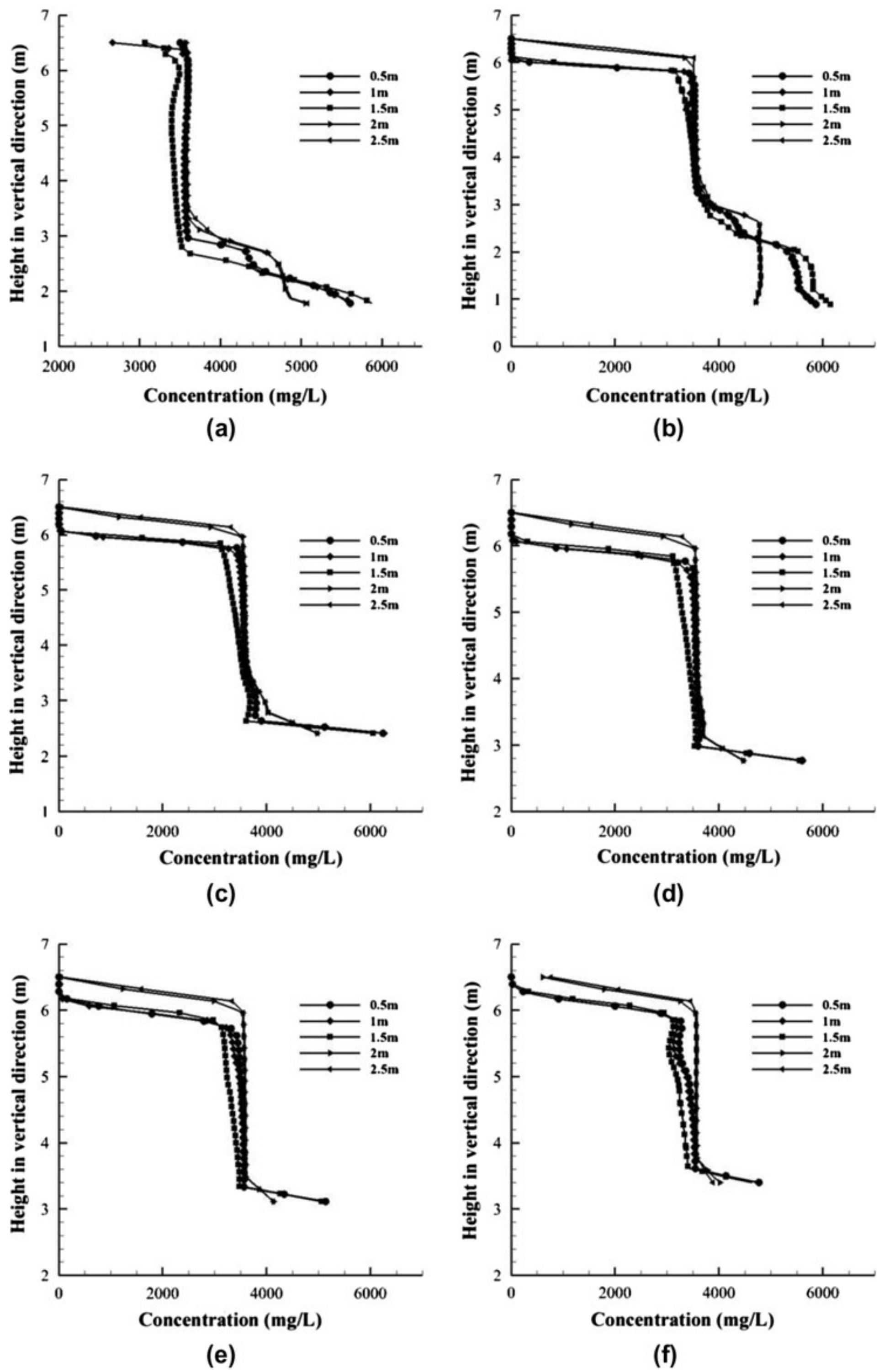


Fig. 4. Vertical distributions of sludge concentration in different sections: (a) $x = 0.2$ m cross section, (b) $x = 4$ m cross section, (c) $x = 10$ m cross section, (d) $x = 15$ m cross section, (e) $x = 20$ m cross section, and (f) $x = 24$ m cross section.

Table 1
Relations of the baffle-horizontal positions, SS values at water outlet, and the removal rates of solids

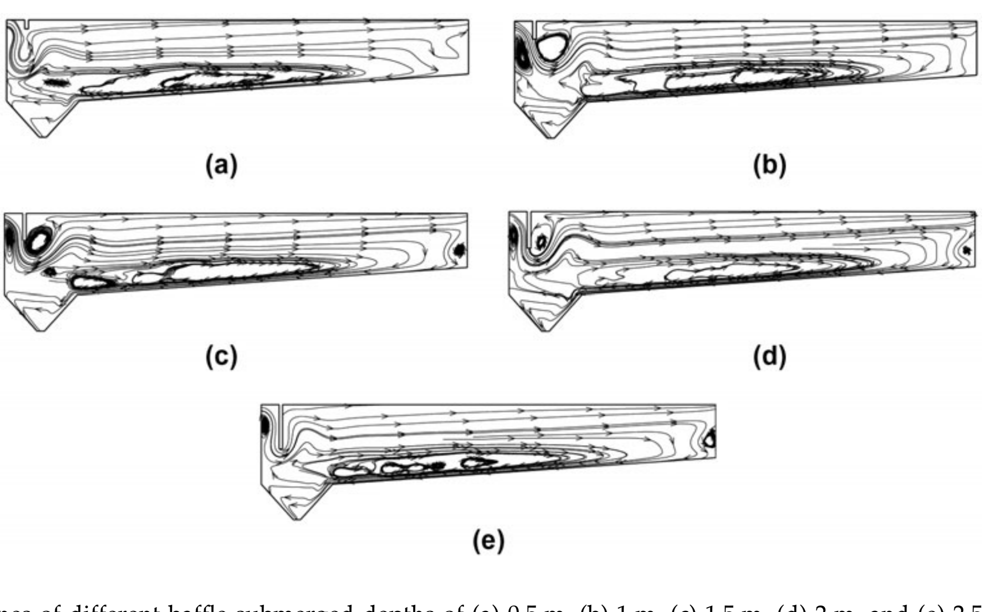
Baffle-horizontal distance away from the water inlet (m)	Difference of solid rate between import and export (kg/s)	SS value at clear water outlet (mg/L)	Removal rate of solids (%)
0.5	4.2112e-06	253.58	92.65
1	2.2397e-06	254.61	92.69
1.5	4.0962e-06	368.46	89.32
2	3.1595e-06	599.27	82.63
2.5	3.7896e-06	710.36	79.41

Among the five horizontal distance of 0.5, 1, 1.5, 2, and 2.5 m away from the water inlet, the removal rate is the highest when the horizontal distances are 0.5 and 1 m, and the removal rates are significantly less when the horizontal distances are of 2 and 2.5 m than the other distances. This is because when the horizontal distance becomes greater, the effect of the baffle-block on the flow becomes not obvious, and so the enhancement of the settling of solids by directing them toward the bottom of the tank becomes poor, which makes the removal rate be small. When the horizontal distances are 0.5 and 1 m, the corresponding removal rates are almost the same, about 92.60%. According to the analysis above, the horizontal baffle-distance is recommended to lie between 0.5 and 2 m.

3.2. Analysis of the effect of different baffle-submerged depths on flow fields and the concentration field distributions

The baffle is set 1 m away from the water inlet. The hydraulic characteristics of the sedimentation basin is studied under the five baffle-submerged depths of 0.5, 1, 1.5, 2, and 2.5 m, respectively. Fig. 5 shows the computed streamlines of different baffle-submerged depths.

Fig. 5(a) shows that, with the baffle submerged at a depth of 0.5 m, upstream and downstream of the baffle no recirculation zone appears, but below the baffle a small recirculation zone exists. Near the bottom of the tank, the flow pattern is complicated, and different sizes of vortex exist. With the baffle-submerged depth increased to 1 m, as shown in Fig. 5(b), upstream and downstream of the baffle a recirculation zone forms respectively, and the recirculation zone downstream of the baffle is larger, and the recirculation zone near the bottom of the tank becomes larger. With the baffle-submerged depth increased to 1.5 m, as shown in Fig. 5(c), upstream and downstream of the baffle the recirculation zones become smaller. A small recirculation zone appears below the water outlet. With the baffle-submerged depth increased to 2 m, as shown in Fig. 5(d), upstream and downstream of the baffle the recirculation zones become even smaller, and the recirculation below



(a) (b) (c) (d) (e)

Fig. 5. Streamlines of different baffle-submerged depths of (a) 0.5 m, (b) 1 m, (c) 1.5 m, (d) 2 m, and (e) 2.5 m.

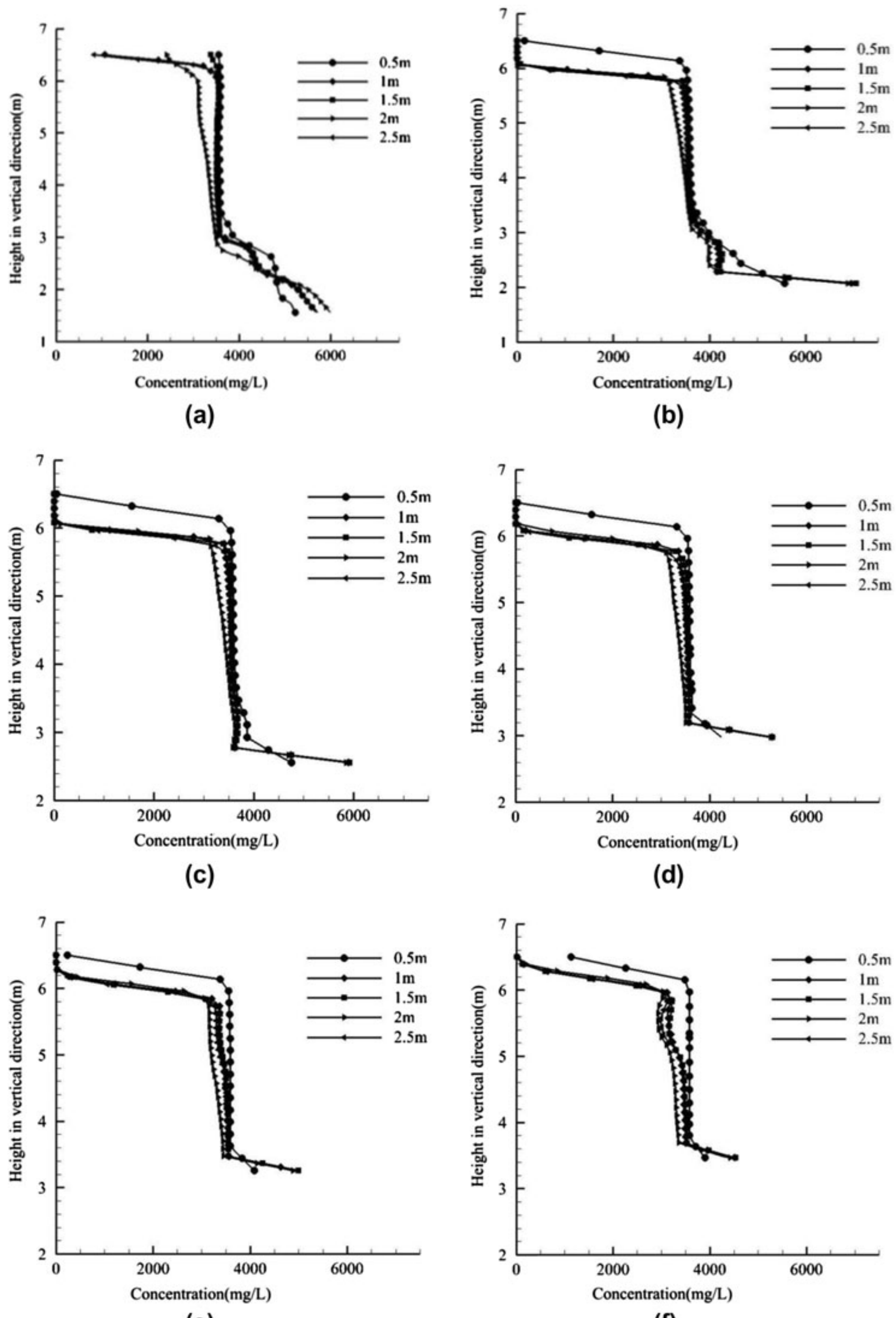


Fig. 6. Vertical distributions of sludge concentration in different sections: (a) cross section of $x = 0.2$ m, (b) cross section of $x = 4$ m, (c) cross section of $x = 10$ m, (d) Cross section of $x = 15$ m, (e) Cross section of $x = 20$ m, and (f) Cross section of $x = 24$ m.

the water outlet remains nearly unchanged .With the baffle-submerged depth increased to 2.5 m, as shown in Fig. 5(e), upstream of the baffle the recirculation zone becomes even smaller, and downstream of the baffle it nearly disappears. The recirculation zone below the water outlet remains still unchanged.

Fig. 6 shows the comparisons of the vertical distributions of the concentrations of different sections,

Table 2
Relations of the baffle-submerged depths, SS values at water outlet, and the removal rates of solids

Baffle-submerged depth (m)	Difference of solid rate between import and export (kg/s)	SS value at water outlet (mg/L)	Removal rate of solids (%)
0.5	4.1592e-06	679.58	74.36
1	5.1583e-06	503.01	85.42
1.5	4.9591e-06	361.56	89.52
2	4.8648e-06	294.28	91.47
2.5	4.7566e-06	299.75	92.01

which indicates that total distribution of the concentration of the sections except that at $x = 0.2$ m decreases with an increase in the baffle-depth. With the baffle-depth of 0.5 m being too small, its effect on water flow is small, which makes the scope of the clear water very small.

Table 2 shows the effect of different baffle-depths on the removal rate of water at outlet. It can be seen that with an increase in baffle-depth from 0.5 to 2 m, the removal rate of outlet increases, and especially, with baffle-depth from 0.5 to 1 m, the removal rate increases obviously. However, with the baffle length from 2 to 2.5 m, the removal rate increases slightly, which shows that the baffle does not need a too big length.

4. Some discussions and further study plan

Numerical simulation and experimental methods depend upon each other. Experiment is the main way to investigate a new basic phenomenon, taking a large amount of observation data as the foundation, still, the validation for a numerical simulation result must use the measured (in prototype or model) data. Doing numerical simulation in advance can give the preliminary results, which can make the corresponding experiment plan more purposeful, and often reduce the number of tests needed by systematically doing experiments, and are very useful for the design of experimental device [11].

Here, a numerical tool has been used to study the effect of a baffle on the hydraulic characteristics in a sedimentation tank. Next, further study will be done to validate the simulation model by an experimental method. An experiment with a 1:10 scale model made of organic glass for the sedimentation tank will be performed according to the gravity similarity theory.

Particle dynamic analyser (PDA) will be used to the test model for the sedimentation tank with different positions of a baffle. Fig. 7(a) shows the experimental system of mixture circulation. Fig. 7(b) shows the sketch of the principle of PDA. PDA (type 58N50) produced by Dantec cooperation, Denmark, is employed to measure the flow in the sedimentation

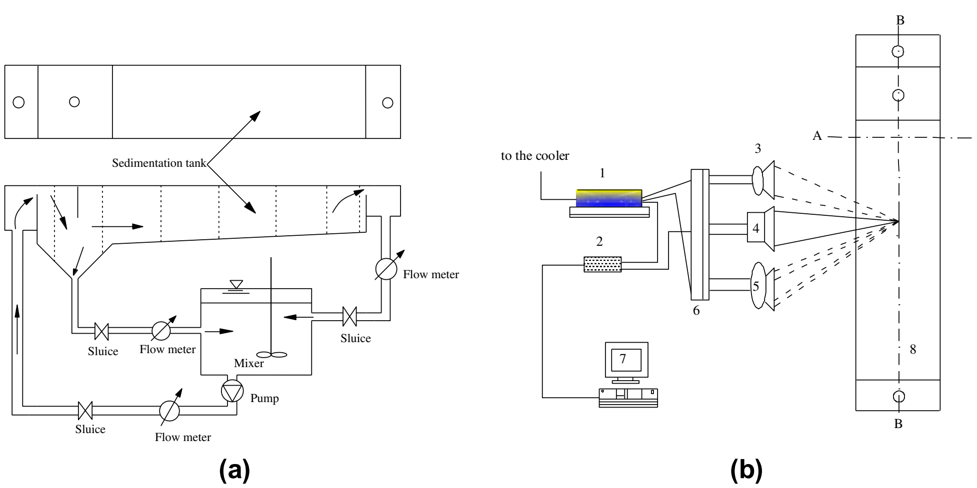


Fig. 7. Sketch of (a) experimental system and (b) measurement of PDA.
Notes: (1) Laser; (2) signal processor; (3) one-dimensional fiber-optic probe; (4) receiver; (5) two-dimensional fiber-optic probe; (6) self-motion shelf; (7) computer; (8) sedimentation tank.

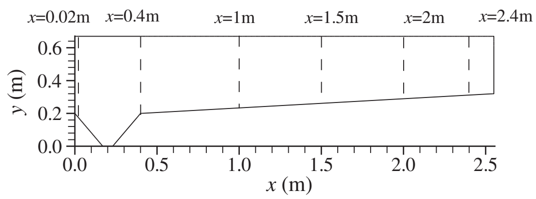


Fig. 8. Vertical lines for the measurement of velocity and sludge concentration in different sections.

tank. It is developed on the basis of traditional laser Doppler anemometry and composed of laser, transmitting system, receiver, signal processor, computer, as well as a three-dimensional self-motion shelf, as shown in Fig. 7(b). The velocity, diameter, and concentration could be got simultaneously without disturbing the flow field. Details of PDA theory can be found in Refs. [12,13]. The different measurement sections along x-axis in the sedimentation tank are illustrated with the dashed lines in Fig. 8.

After measuring the velocity values, the corresponding values in the prototype can be computed, and the measured velocity values can be compared with calculated data, and further, the reliability of the calculation will be verified. After validating the simulation model, it will be used to predict the flow fields in a sedimentation tank with different installing positions and submerged depths of a baffle, and the predicted results are used to obtain the optimal installing position and submerged depth of the baffle. Finally, the corresponding installing position and submerged depth of the baffle will be obtained in the prototype according to the hydraulic similarity theory.

5. Conclusions

The baffle has a great effect on the hydraulic characteristics and solid removal rate in a sedimentation tank, and thus, a proper baffle including its horizontal position and submerged depth should be used. By the numerical simulation for the sedimentation tank, it was found that: (1) with the horizontal baffle-distance from 0.5 to 2.5 m, the smaller the horizontal baffle-distance is, the bigger is the removal rate. The removal rate ranges from 82.63 to 92.6% with the horizontal baffle-distance between 0.5 and 2 m. Therefore, the horizontal baffle-distance is recommended to lie between 0.5 and 2 m; and (2) with an increase in baffle-depth being from 0.5 to 2 m, the removal rate at outlet increases, and especially, with the baffle-depth being from 0.5 to 1 m, the removal rate increases obviously. However, with the baffle length being from 2 to 2.5 m, the removal rate decreases slightly, which shows that the baffle does not need a too big length, and 2 m is recommended as the proper battle length.

In the actual project, due to the complexity of turbulence, the effect of a baffle on the hydraulic characteristics and solid removal rate in the sedimentation tank also needs a further research with an experimental method to validate the simulation results.

Acknowledgments

Financial support of this study came from the National Natural Science Foundation of China (Grant Nos. 51578452 and 51178391), the Scientific research project of Shaanxi Province (2014K15-03-05) and special funds for the development of characteristic key disciplines in the local university supported by the Central Financial Fund (Grant No: 106-00X101) were greatly appreciated.

References

[1] A.M. Goula, M. Kostoglou, T.D. Karapantsios, A.I. Zouboulis, CFD methodology for the design of sedimentation tanks in potable water treatment. Case study: The influence of a feed flow control baffle, Chem. Eng. J. 140 (2007) 110–121.
[2] G.A. Ekama, P. Marais, Assessing the applicability of the 1D flux theory to full-scale secondary settling tank design with a 2D hydrodynamic model, Water Res. 38 (3) (2004) 495–506.
[3] S. Sarkar, D. Kamilya, B.C. Mal, Effect of geometric and process variables on the performance of inclined plate settlers in treating aquacultural waste, Water Res. 41 (2007) 993–1000.
[4] A.M. Razmi, B. Firoozabadi, G. Ahmadi, Experimental and numerical approach to enlargement of performance of primary settling tanks, J. Appl. Fluid Mech. 2 (2008) 1–13.
[5] L. Fan, N. Xu, X. Ke, H. Shi, Numerical simulation of secondary sedimentation tank for urban wastewater, J. Chin. Inst. Chem. Eng. 38 (2007) 425–433.
[6] E. Ramin, D.S. Wágner, L. Yde, P.J. Binning, M.R. Rasmussen, P.S. Mikkelsen, B.G. Plósz, A new settling velocity model to describe secondary sedimentation, Water Res. 66 (2014) 447–458.
[7] E. Doroodchi, K.P. Galvin, D.F. Fletcher, The influence of inclined plates on expansion behaviour of solid suspensions in a liquid fluidised bed—A computational fluid dynamics study, Powder Technol. 156 (2005) 1–7.
[8] R. Amini, R. Taghipour, H. Mirgolbabaei, Numerical assessment of hydrodynamic characteristics in chlorine contact tank, Int. J. Numer. Methods Fluids 67 (2011) 885–898.
[9] W.L. Wei, Y.L. Liu, Theory of Multiphase Flow Simulation With Applications in Wastewater Treatment Engineering, first ed., Xi'an Shanxi Science and Technology Publishing House, Xi'an, 2015, 95–99.

[10] Y.L. Liu, P. Zhang, W.L. Wei, 2D simulation of effects of position of baffles on the removal rate of solids in a sedimentation tank, Appl. Mech. Mater. 253–255 (2013) 861–864.
[11] W. Wei, Y. Liu, B. Lv, Numerical simulation of optimal submergence depth of impellers in an oxidation ditch, Desalin. Water Treat. 13 (2015) 1–8, doi: 10.1080/19443994.2015.1021840.
[12] W.D. Bachalo, Method for measuring the size and velocity of spheres by dual beam light scatter interferometry, Appl. Opt. 19 (3) (1980) 363–370.
[13] M. Rashidi, G. Hetsroni, S. Banerjee, Particle–turbulence interaction in a boundary layer, Int. J. Multiphase Flow 16(6) (1990) 935–949.