Numerical Investigation of Baffle Effect on the Flow in a Rectangular Primary Sedimentation Tank

M. Shahrokhi, F. Rostami, M.A. Md Said, S. Syafalni

Abstract—It is essential to have a uniform and calm flow field for a settling tank to have high performance. In general, the recirculation zones always occurred in sedimentation tanks. The presence of these regions may have different effects. The non-uniformity of the velocity field, the short-circuiting at the surface and the motion of the jet at the bed of the tank that occurs because of the recirculation in the sedimentation layer, are affected by the geometry of the tank. There are some ways to decrease the size of these dead zones, which would increase the performance. One of the ways is to use a suitable baffle configuration. In this study, the presence of baffle with different position has been investigated by a finite volume method, with VOF (Volume of Fluid) model. Besides, the k-ε turbulence model is used in the numerical calculations. The results indicate that the best position of the baffle is obtained when the volume of the recirculation region is minimized or is divided to smaller part and the flow field trend to be uniform in the settling zone.

Keywords—Sedimentation tanks, Baffle, Numerical Modeling, VOF, Circulation Zone

I. INTRODUCTION

THE removal of suspended and colloidal materials from water and wastewater by gravity separation is one of the most widely used unit operations in water and wastewater treatment. Sedimentation is the separation of suspended particles that are heavier than water; the separation is achieved through gravitational settling. Two types of equipment are used in the sedimentation process: the grit chamber (plain sedimentation) and the sedimentation tank (clarifier).

The two main types of sedimentation tanks are primary and secondary or final settling tanks. A primary settling tank has low influent concentration. Its flow field is minimally influenced by the concentration field, and its buoyancy effects can be negligible. Secondary or final settling tanks, however, have higher influent concentration.

A uniform flow field is essential to increase the efficient performance of settling tank. This enables particles to settle at a constant velocity and in less time. The existence of

M. Shahrokhi is with the Universiti Sains Malaysia, P.Penang, Malaysia (e-mail: mshotm2000@yahoo.com).

F. Rostami is with the Universiti Sains Malaysia, P.Penang, Malaysia (e-mail: fa.rostami@gmail.com).

M.A. Md Said is with the Universiti Sains Malaysia, P.Penang, Malaysia (e-mail: azlin@eng.usm.my).

S. Syafalni is with the Universiti Sains Malaysia, P.Penang, Malaysia (e-mail: cesyafalni@eng.usm.my).

circulation regions in the tank may have different effects. These dead zones decrease the effective volume of the tanks that may result in a short circuit between the inlet and outlet of the tank. Some part of the flow exits the tank without any settling. It also induces high turbulence intensity in certain regions, which not only decreases the possibility of particle deposition, but may also cause resuspension problems.

Density currents, circulation, and short circuiting are hydraulic phenomena that occur in sedimentation tanks because of density differences, stratification of liquid solids, and tank geometry [1]. Transverse baffle can reduce the effects of these factors, and enhance sedimentation performance [2].

Crosby [3] observed that a mid-radius baffle extending from the floor up to mid-depth decreased the effluent SS concentration of the clarifier by 37.5%. Velocity and concentration fields for a rectangular clarifier equipped with an intermediate baffle showed that the installation of an intermediate baffle was an effective approach [4]. Zhou et al. [5] applied numerical modeling in studying the performance of circular secondary clarifiers with reaction baffles under varying solid and hydraulic loadings. The importance of a baffle in dissipating the kinetic energy of incoming flow and reducing short circuiting indicates that the location of the baffle has a profound effect on the nature of the flow [5].

Huggins et al., [6] who tested a number of potential raceway design modifications, noticed that by adding a baffle, the overall percentage of solid removal efficiency increased from 81.8% to 91.1%. Fan et al. [7] observed that the solid concentration profile in the flow region near the baffle is similar to that obtained without a baffle. By contrast, solid concentration increases sharply in the outer region of the baffle, which suggests that the solid phase congregates rapidly at the end of the baffle. Tamayol et al. [8] found that the best position for the baffle is somewhere in the circulation zone to spoil this circulation region.

Goula et al. [9] used numerical modeling to study particle settling in a sedimentation tank equipped with a vertical baffle installed at the inlet. The authors showed that the baffle increased particle settling efficiency from 90.4% for a standard tank without a baffle to 98.6% for a tank with an installed baffle. Installing baffles improves the performance of a tank in terms of settling. The baffles act as barriers,

World Academy of Science, Engineering and Technology
International Journal of Environmental and Ecological Engineering
Vol:5, No:10, 2011

effectively suppressing the horizontal velocities of the flow and forcing the particles to the bottom of the basin [10].

The main objective of this study is to determine the optimal position of baffles in a settling tank. Because comprehensive standards are not available for the design of baffle positions, the best baffle location is determined through numerical methods.

II. GOVERNING EQUATION

A. Time-Averaged Flow Equations

The governing equations are general mass continuity and momentum. The turbulence model is also solved with these equations to calculate the Reynolds stresses. The mass continuity equation for fluids is simple. The flow pattern is assumed to be two-dimensional, enabling the calculation of two momentum equations in the x and z directions, as well as the length and height of the tank. The general mass continuity equation is [11-12].

$$\frac{\partial}{\partial x}\left(uA_x\right) + \frac{\partial}{\partial z}\left(wA_z\right) = 0 \tag{1}$$

where V_f is the fractional volume of flow in the calculation cell; ρ is the fluid density; and (u,w) are the velocity components in the length and height (x, z). The momentum equation for the fluid velocity components in the two directions are the Navier–Stokes equations, expressed as follows:

$$\frac{\partial u}{\partial t} + \frac{1}{V_f}\left\{uA_x\frac{\partial u}{\partial x} + wA_z\frac{\partial u}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + G_x + f_x \tag{2}$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_f}\left\{uA_x\frac{\partial w}{\partial x} + wA_z\frac{\partial w}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial p}{\partial z} + G_z + f_z \tag{3}$$

where G_x, G_z are body accelerations, and f_x, f_z are viscous accelerations. Variable dynamic viscosity μ are as follows:

$$\rho V_f f_x = -\left\{\frac{\partial}{\partial x}\left(A_x\tau_{xx}\right) + \frac{\partial}{\partial z}\left(A_z\tau_{xz}\right)\right\} \tag{4}$$

$$\rho V_f f_z = -\left\{\frac{\partial}{\partial x}\left(A_x\tau_{xz}\right) + \frac{\partial}{\partial z}\left(A_z\tau_{zz}\right)\right\} \tag{5}$$

where

$$\tau_{xx} = -2\mu\frac{\partial u}{\partial x} \tag{6}$$

$$\tau_{zz} = -2\mu\frac{\partial w}{\partial z} \tag{7}$$

$$\tau_{xz} = -\mu\left\{\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right\} \tag{8}$$

Fluid surface shape is illustrated by volume of fluid (VOF) function F(x, z, t). With the VOF method, grid cells are classified as empty, full, or partially filled with fluid. Cells are allocated in the fluid fraction varying from zero to one, depending on fluid quantity. Thus, in F=1, fluid exists, whereas F=0 corresponds to a void region. This function displays the VOF per unit volume and satisfies the equation [11].

$$\frac{\partial F}{\partial t} + \frac{1}{V_f}\left\{\frac{\partial}{\partial x}\left(FA_x u\right) + \frac{\partial}{\partial z}\left(FA_z w\right)\right\} = 0 \tag{9}$$

F in one phase problem depicts the volume fraction filled by the fluid. Voids are regions without fluid mass that have a uniform pressure appointed to them. Physically, they represent regions filled with vapor or gas, whose density is insignificant in relation to fluid density.

B. k-ε Turbulent Model

The simplest model consists of a transport equation for the specific kinetic energy associated with turbulent velocity fluctuations and a parameter that characterizes some other property of the turbulence. The choice of parameters is arbitrary, provided it can be used with the kinetic energy to determine the length and time scales characterizing the turbulence.

A slightly more sophisticated (and more widely used) model is made up of two transport equations for turbulent kinetic energy k and its dissipation ε; this is the so-called k-ε model [13]. The k-ε model provides reasonable approximations of many types of flows, although it sometimes requires modification of its dimensionless parameters (or even functional changes to terms in the equations) [14]. The turbulence kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations:

$$\frac{\partial k}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial k}{\partial x} + vA_y\frac{\partial k}{\partial y} + wA_z\frac{\partial k}{\partial z}\right\} = \tag{10}$$

$$P + G + Diff - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + \frac{1}{V_F}\left\{uA_x\frac{\partial \varepsilon}{\partial x} + vA_y\frac{\partial \varepsilon}{\partial y} + wA_z\frac{\partial \varepsilon}{\partial z}\right\} = \tag{11}$$

$$\frac{C_{1\varepsilon}.\varepsilon}{k}\left(P + C_{3\varepsilon}.G\right) + DDif - C_{2\varepsilon}.\frac{\varepsilon^2}{k}$$

where P is shear production, G is buoyancy production, *Diff* and *DDif* represent diffusion, and $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ are constant. In a standard k-ε model, $C_{1\varepsilon}$=1.44 and $C_{2\varepsilon}$=1.92.

In this paper, a module of FLOW-3D® flow solver (version 9.4), which utilizes a finite volume (or finite difference)

World Academy of Science, Engineering and Technology
International Journal of Environmental and Ecological Engineering
Vol:5, No:10, 2011

scheme for structured meshes, is used to simulate the free surface flow in these tanks. The flow field is separated into fixed rectangular cells. The local average values of all dependent variables for each cell are computed. Pressures and velocities are associated implicitly by using time-advanced pressures in momentum equations and time-advanced velocities in the mass (continuity) equation. These semi-implicit formulations of the finite-difference equations enable the efficient resolution of low speed and incompressible flow problems. The semi-implicit formulation, however, results in coupled sets of equations that must be solved by an iterative technique [15].

III. NUMERICAL MODEL

The velocity profiles achieved from the numerical method, in comparison with the experimental results [16], are shown in the Fig. 1. In this figure, the solid lines are the numerical results. The numerical data excellently match the laboratory results, but some errors are observed near the surface and close to the bed. These errors can be attributed to high sediment concentration at the bottom of the tank that was neglected in the numerical simulation.
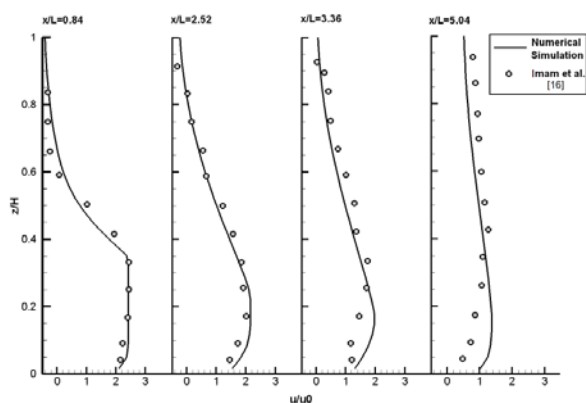

Fig. 1 Comparison of velocity distribution from present work with experimental result [16]

Steady state incompressible flow conditions with viscose effect are generally considered in hydraulic numerical modeling, and the Navier–Stokes equation has been well-verified as an effective solution to the governing equation. The Navier–Stokes equation is an incompressible form of the conservation of mass and momentum equations, and is comprised of non-linear advection, rate of change, diffusion, and source term in the partial differential equation. The mass and momentum equations joined by velocity can be used to obtain an equation for the pressure term. When the flow field is turbulent, computation becomes more complex. Because of this, the Reynolds-Averaged Navier–Stokes (RANS) equation is prevalently used. It is a modified form of the Navier–Stokes equation and includes the Reynolds stress term, which approximates the random turbulent fluctuations by statistics.

In this study, the computational fluid dynamics (CFD) program FLOW-3D®, developed by Flow Sciences, was used for the numerical simulation. FLOW-3D® solves the RANS equations by the finite volume formulation gained from a rectangular finite difference grid. For each cell, mean values of the flow parameters, such as pressure and velocity, are calculated at discrete times. The new velocity in each cell is computed from the coupled momentum and continuity equation using previous time step values in each of the centers of the cell faces. The pressure term is obtained and adjusted using the estimated velocity to satisfy the continuity equation. With the computed velocity and pressure for a later period, the remaining variables are estimated involving turbulent transport, density advection and diffusion, and wall function evaluation.

We used the Fractional Area/Volume Obstacle Representation (FAVOR) method to inspect the geometry in the finite volume mesh [12]. FAVOR appoints the obstacles in a calculation cell with a factional value between zero to one as obstacle fills in the cell. The geometry of the obstacle is placed in the mesh by setting the area fractions on the cell faces along with the volume fraction open to flow [17]. This approach creates an independent geometry structure on the grid, and then the complex obstacle can be produced.

IV. GEOMETRY SPECIFICATION

Flow field in sedimentation tanks is three-dimensional. The degree of importance of the three-dimensional effects is related to the location of the inlet and outlet of a basin and their widths. The inlet and outlet are assumed to uniformly spread the width of the basins, making the three-dimensional effects negligible. For simplicity, two-dimensional models were used for our simulations.
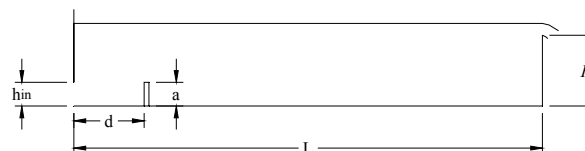

Fig. 2 Schematic diagram of the tank

The geometry of the longitudinal sedimentation basin with baffles is illustrated in Fig. 2. The basin is 200 cm long, 50 cm deep and 50 cm wide. A weir is located at the end of the basin to regulate the flow height of H=30 cm. Baffle height a=5.5 cm. The inlet flow goes through a sluice gate with an opening of h_{in}=10 cm. Fig. 2 shows the baffle distance from the inlet of the tank and the flow rate equal to 2 L/s. Extensive tests were conducted to establish a grid-independent solution. More than 288×69 mesh points were required before the velocity contours transformed into an independent grid; then, 288×69 grids were chosen for the computation. Thus, the mesh with approximately 19872 cells was used. The boundary condition for the inlet flow (influent) is specified velocity, and that for the outflow (effluent) is outflow. No slip conditions were applied at the rigid walls, and these were treated as nonpenetrative boundaries. Free surface boundary was calculated by the VOF method. No slip conditions for velocity and standard wall functions were used for turbulence modeling.

World Academy of Science, Engineering and Technology
International Journal of Environmental and Ecological Engineering
Vol:5, No:10, 2011

TABLE I
CIRCULATION VOLUME PERCENTAGE IN DIFFERENT LOCATION OF THE BAFFLE

	No Baffle	Ratio of the baffle distance from inlet of the tank with length of tank (d/L)							
		0.120	0.125	0.135	0.150	0.200	0.250	0.300	0.400
Circulation Volume (%)	37.05	34.62	32.28	34.10	34.40	34.43	35.08	35.50	37.78

d: The baffle distance from the inlet of the tank; L: The length of the tank

V. RESULTS AND DISCUSSION

Three circulation regions may appear with size sensitive to the position of the baffle when a baffle is used in the tank. The best position for the baffle is obtained when the volume of the circulation zone is minimized or the dead zone is divided into smaller parts. Thus, the best position for the baffle may lead to a more uniform distribution of velocity in the tank and minimize dead zones. Small recirculation zones, which are important to sedimentation, are also found near the entry and exit weir.

because of the size of the dead zone. Table I show that the baffle position at d/L=0.125 exhibits the best performance.

Furthermore, Table I illustrates that with increasing baffle distance from point d/L=0.125, the volume of the dead zone gradually increases. Consequently, the removal efficiency of the tank also decreases.

Fig. 3 shows the streamline of different baffle locations in the sedimentation tank. Two circulation zones exist in the tank at d/L=0.125. The circulation volume, however, remains minimized and the baffle presumably separates the dead zone into two sections.
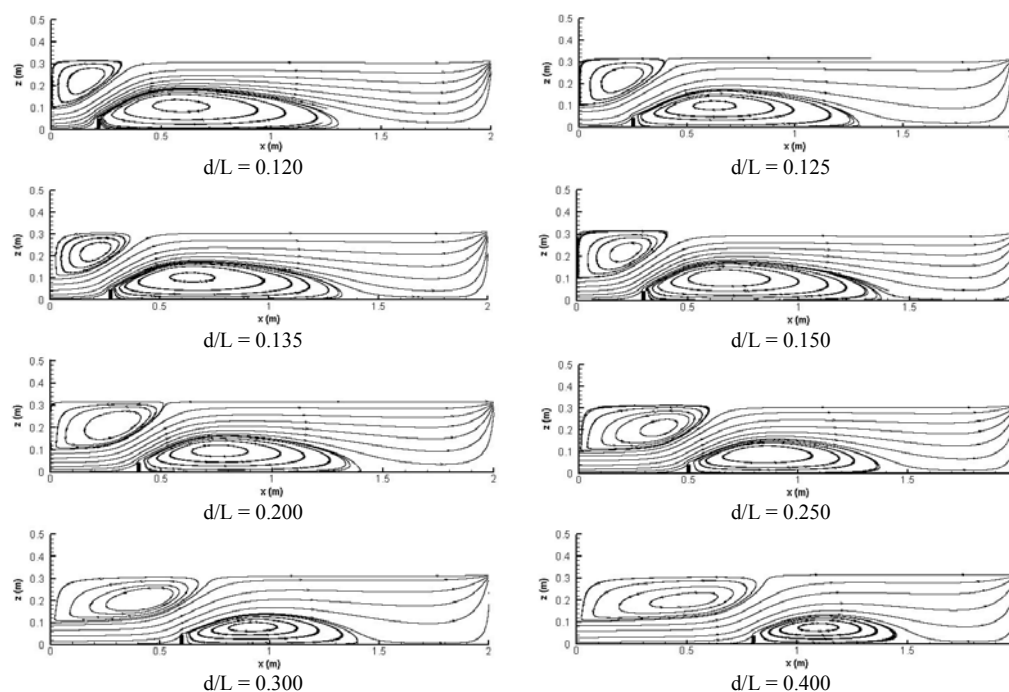

d/L = 0.120   d/L = 0.125

d/L = 0.135   d/L = 0.150

d/L = 0.200   d/L = 0.250

d/L = 0.300   d/L = 0.400

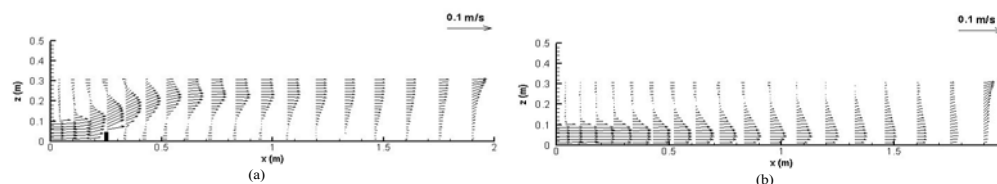Fig. 3 Streamlines of different baffle positions in sedimentation tank

Fig. 4 x-velocity vector from above a) Baffle at d/L= 0.125, b) No baffle

Different baffle positions were modelled in this study. Circulation volume, which is normalized by the total water volume in the tank and calculated by the numerical method, is shown in Table I. The table indicates the absolute predictability of some cases to exhibit weak performance

Fig. 4 indicates the X-velocity profiles in a no-baffle tank and the tank in which the baffle is at the optimum position. The comparison between these two profiles shows that the X-velocity after the baffle was installed is smaller than that in the tank in which no baffle was used. Particularly in the bed and after baffle location was modified, the Z-velocity changes to a

World Academy of Science, Engineering and Technology
International Journal of Environmental and Ecological Engineering
Vol:5, No:10, 2011

downward direction, resulting in significantly enhanced sedimentation.

Turbulent kinetic energy k in several cases of flow is shown in Table II This parameter has a range of values for different baffle locations. Table II shows decreasing turbulent kinetic energy for the optimal baffle case (d/L=0.125). The comparison between cases (a) and (b) in Fig. 5 shows that using the baffle in the settling basin causes the kinetic energy to decrease near the bed and the zone with high kinetic energy moves to the upper region of the basin. The baffle creates a region with low amounts of kinetic energy near the bed. The ability of flow to carry the sediment is not significant and the sedimentation process may increase.

TABLE II
TURBULENT KINETIC ENERGY k IN x/L

d/L	x/L		
	0.4	0.6	0.8
0.120	1.43E-04	1.25E-04	9.5E-05
0.125	1.72E-04	1.19E-04	9.1E-05
0.135	1.78E-04	1.26E-04	9.5E-05
0.150	1.80E-04	1.27E-04	9.5E-05
0.200	1.76E-04	1.29E-04	9.4E-05
0.250	1.68E-04	1.30E-04	9.2E-05
0.300	1.68E-04	1.29E-04	9.2E-05
0.400	2.08E-04	1.40E-04	9.4E-05
No Baffle	1.95E-04	1.36E-04	9.7E-05

d: The baffle distance from the inlet of the tank;
L : The length of the tank;
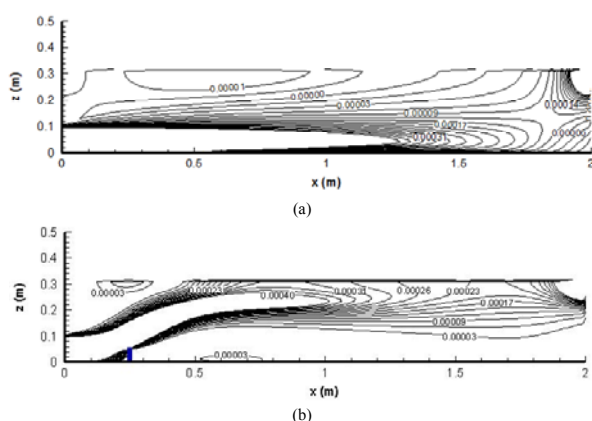x : The distance from the inlet of the tank

(a)

(b)

Fig. 5 Contour of kinetic energy a) No baffle, b) Baffle at d/L= 0.125

VI. CONCLUSION

Sedimentation by gravity is one of the most common and extensively applied techniques in the removal of suspended solids from water and wastewater. Investment in settling tanks accounts for about 30% of the total investment in a treatment plant [8]. The calculation of sedimentation performance has been the subject of numerous theoretical and experimental studies. Sedimentation performance depends on the characteristics of the suspended solids and flow field in the tank. A uniform and calm flow field is essential for a tank to have high efficiency. This facilitates particle deposition at a constant velocity in lesser time. In general, circulation regions are always present in settling tanks. Circulation zones are named dead zones, because water is trapped and particulate fluid will have less volume for flow and sedimentation in these regions. The existence of large circulation regions, therefore, will lowers tank efficiency.

Moreover, the formation of circulation zones diminishes the performance of the sedimentation tank by short circuiting, and positioning a baffle in an appropriate location can reduce the formation of these zones. This means that correctly positioning a baffle prevents the formation of the bottom jet moving to the surface of the basin and spilling over at the outlet.

Numerical approaches were carried out to investigate the effects of baffle location on the flow field. Using CFD and VOF methods, we developed a numerical simulation of flow in the tank through the FLOW-3D® software. Results show that the installation of a baffle improves tank efficiency in terms of sedimentation. The baffle acts as a barrier, effectively suppressing the horizontal velocities of the flow and reducing the size of the dead zones. A baffle also reduces turbulent kinetic energy and induces a decrease in maximum magnitude of the stream-wise velocity and upward inclination of the velocity field compared with the no-baffle tank. On the basis of these results, we conclude that the baffle must be placed near the circulation region.

REFERENCES

[1] Silveston, P.L., J.F. Cordoba-Molina, and R.R. Hudgins, "The use of flow contraction to improve clarifier performance". *Water Science and Technology*, 13 1981: p. 385-394.
[2] Hamlan, M.J. and A.H.A. Wahab, "Settling Characteristics of Sewage in Density Currents". *Water Research*, 4 1970: p. 251-271.
[3] Crosby, R.M., *Evaluation of the Hydraulic characteristics of activated sludge secondary clarifiers*. 1984, Environmental Protection Agency, Office of Research and Development: Washington, D.C. U.S. p. 84-131.
[4] Bretscher, U., P. Krebs, and W.H. Hager, "Improvement of flow in final settling tanks". *J. Enviromental Engineering, ASCE*, 118 1992.(3): p. 307-321.
[5] Zhou, S., J. McCorquodale, and Z. Vitasovic, "Influences of density on circular clarifiers with baffles". *Journal of Environmental Engineering, ASCE*, 118 1992.(6): p. 829-847.
[6] Huggins, D.L., R.H. Piedrahita, and T. Rumsey, "Analysis of sediment transport modeling using computational fluid dynamics (CFD) for aquaculture raceways". *Aquacult. Eng.*, 31 2005: p. 277-293.
[7] Fan, L., et al., "Numerical simulation of secondary sedimentation tank for urban wastewater". *J. Chin. Inst. Chem. Eng.*, 38 2007: p. 425-433.
[8] Tamayol, A., B. Firoozabadi, and G. Ahmadi, "Effects of Inlet Position and Baffle Configuration on Hydraulic Performance of Primary Settling Tanks". *Journal of Hydraulic Engineering, ASCE.*, 134 2008.(7): p. 1004-1009.
[9] Goula, A.M., et al., " A 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: p. 110-121.
[10] Sammarraee, M.A. and A. Chan, "Large-eddy simulations of particle sedimentation in a longitudinal sedimentation basin of a water treatment

World Academy of Science, Engineering and Technology
International Journal of Environmental and Ecological Engineering
Vol:5, No:10, 2011

plant. Part 2: The effects of baffles*". Chemical Engineering Journal* 152 2009: p. 315-321.

[11] Hirt, C.W. and B.D. Nichols, "Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries*". J. Comp. Phys.*, 39 1981: p. 201-225.

[12] Hirt, C.W. and J.M. Sicilian, *A Porosity Technique for the Definition of Obstacles in Rectangular Cell Meshes*, in *Fourth International Conf. Ship Hydro*. 1985, National Academy of Science: Washington, DC. p. 1-19.

[13] Harlow, F.H. and P.I. Nakayama, "Turbulence Transport Equations*". Phys. of Fluids*, 10 1967.(11): p. 2323-2333.

[14] Svendsen, I. and J. Kirby, *Numerical study of a turbulent hydraulic jump*, in *17th ASCE Engineering Mechanics Conference*. 2004: University of Delaware, Newmark, DE.

[15] FlowScience, *Flow-3D user manual*. 2009.

[16] Imam, E., J.A. McCorquodale, and J.K. Bewtra, "Numerical Modeling of Sedimentation Tanks*". Journal of Hydraulic Engineering, ASCE.*, 109 1983.(12): p. 1740-1754.

[17] Hirt, C.W., *Identification and Treatment of Stiff Bubble Problems*. 1992, Flow Science Inc.