Membrane Water Treatment, *Vol. 8, No. 5 (2017) 423-435*
DOI: https://doi.org/10.12989/mwt.2017.8.5.423

Performance and flow field assessment of settling tanks using experimental and CFD modeling

Alireza Zamani Nouri*[1] and Mohammad Mehdi Heydari[2]

[1]*Department of Civil Engineering, College of Engineering, Saveh Branch, Islamic Azad University, Saveh, Iran*
[2]*Kahana Water and Wastewater Company, Iran*

(*Received January 18, 2017, Revised May 2, 2017, Accepted May 4, 2017*)

Abstract. Settling Basins are one of the most important and popular methods for removal of suspended sediments irrigation and drainage networks or power canals taking off from an alluvial river and wastewater treatment plant. Improving the performance and so increasing sediment removal efficiency of settling basins by an alternative method is necessary. In the present work, the effect of baffle and its angle of attack with the flow (θ) on the sediment removal efficiency is investigated by conducting a series of experiments on a straight canal with 8 m length, 0.3 m width and 0.5 m height and 3 m length of basin equipped with an adjustable glass baffle. A numerical analysis has been carried out using ANSYS Fluent 3D software (a general purpose computational fluid dynamics simulation tool) for three Froude numbers from the experiments. The numerical and experimental results were found to match reasonably well.

Keywords: baffle angle; removal efficiency; sedimentation basins; suspended sediment

1. Introduction

Sedimentation by gravity is the most common and extensively applied treatment process for the removal of solids from water and wastewater and it has been used for over one hundred years. Sedimentation tanks are one of the major parts of a treatment plant, especially in purification of turbid flows. Finding new and useful methods to increase hydraulic efficiency are the objective of many theoretical, experimental, and numerical studies. In rectangular tanks, the influent enters the basin at the inlet. Energy dissipation is the main objective in designing a primary clarifier inlet. Energy of the influent must be dissipated at the inlet zone by selecting the best position and configuration of the inlet or using the baffles in the inlet zone (Kerbs *et al*. 1995). The two main types of sedimentation (clarifier) tanks are primary and secondary (or final) settling tanks. In the present study, focus is made on the sedimentation basin in irrigation network. One problem in irrigation structures is sedimentation control at the main entrance to the irrigation network (Shetab-Boushehri *et al*. 2010). Every large network of irrigation canals requires at least a proper sedimentation basin. Sedimentation basins are essential hydraulic structures which have to be designed and constructed at all river water intakes to remove most of suspended sediments which

∗Corresponding author, Ph.D., E-mail: Dr.zamani.ar@gmail.com

http://www.techno-press.org/?journal=mwt&subpage=7 ISSN: 2005-8624 (Print), 2092-7037 (Online)

enters the intake by flowing water. A sedimentation basin consists of an oversized section of a canal, built downstream from the canal head works, and its design is based on increasing the canal surface area to reduce the flow velocity low enough to permit much of the fine suspended particles that might otherwise be transported through the canal and be deposited (Vanoni 1975, Vittal and Raghav 1997, Ranga Raju *et al*. 1999). The settled sediments can be removed by mechanical means or flushing. The bigger the basin, the better the retardation of the sediments, but Expenses are higher, dredging must be done more frequently. Therefore, improvement of performance and increasing sediment removal efficiency of sedimentation basins by an alternative method is necessary. The sedimentation performance depends on the characteristics of the suspended solids and the flow field in the basin.

A common approach for increasing sedimentation basins performance is to use baffles (Tamayol *et al*. 2010). Baffles can interrupt short-circuiting, giving rise to a modified flow field and, potentially, improve the basin performance. Energy of influent must be dissipated at the inlet zone by selecting the best position and configuration of the inlet or by using baffles in the inlet zone (Zhou *et al*. 1992). Baffles are usually placed in the front of inlet opening or built at the bottom of the tank to increase their sedimentation performance (Tamayol *et al*. 2010). A uniform flow field is essential to the efficient performance of sedimentation basin. Baffles enable particles to settle at a constant velocity and in a short period of time. Also the circulation zones between the inlet and outlet of the basin decrease and enhance sedimentation performance. Baffle positioning is essential in dissipating the kinetic energy of incoming flow and reducing chances for occurrence of short circuits (Zhou *et al*. 1992). The baffles act as barriers, effectively suppressing the horizontal velocities of the flow and forcing the particles to the bottom of the basin (Shahrokhi *et al*. 2011a). It must be noted that using baffles without enough caution can worsen performance compared with the tank without a baffle.

Most previous studies have been conducted in primary and secondary tanks (settling tanks). There are a number of comprehensive studies on baffled tanks that investigate their hydraulic efficiency (Tamayol *et al*. 2010, McCorquodale and Zhou 1993, Xanthos *et al*. 2010).

Wills and Davis (1962) have studied the effects of transverse and longitudinal baffles (θ=90°) on the performance of the sedimentation tanks and have shown that the transverse baffles decrease short circuiting. Crosby (1984) observed that a mid-radius vertical baffle extending from the floor up to mid-depth decreased the effluent suspended sediments concentration of the clarifier by 37.5%. Krebs *et al* (1992, 1995) and Krebs (1995) investigated the effect of inlet and intermediate vertical baffles on the flow field in final clarifiers. Their research was based on experiments, numerical modeling and analytical relations. Energy dissipation is the main objective in designing a primary clarifier inlet. Energy of influent must be dissipated at the inlet zone by selecting the best position and configuration of the inlet or by using the baffles in the inlet zone (Krebs 1995). Bretscher *et al*. (1992) showed that installation of the intermediate vertical baffle was effective on the velocity and concentration fields for a rectangular settling tank. Ahmed *et al*. (1996) studied the effects of the position and height of the baffle (θ=90°) in a secondary sedimentation tank by the bottom inlet by placing the baffle at three different positions and various heights, qualitatively. The best result was for the case in which an inlet baffle with a height 67% of the total depth was placed in the first 5% of the channel. After testing many potential raceway design modifications, Huggins *et al*. (2005) noticed that by adding a vertical bottom baffle, the overall percentage of solids removal efficiency increased from 81.8% to 91.1%, resulting in a reduction of approximately 51% in the effluent solids. Tamayol (2005) showed that the best position for the inlet is near the bottom and that the existence of a reflection entrance baffle near the free surface of settling tanks can increase the performance of primary settling tanks. Goula *et al*. (2007) used numerical modeling to study particle settling in a sedimentation tank equipped with a vertical baffle installed at the inlet zone (bottom inlet). They 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 bottom baffle. Razmi *et al*. (2008) found that best location of the vertical baffle is obtained when the volume of the circulation zone is minimized or the dead zone is divided into smaller parts, and they showed that this baffle can reduce the size of the dead zones and turbulent kinetic energy in comparison with the no-baffle condition. Shahrohki *et al*. (2011b, 2011c) was performed numerical simulation to investigate the effects of vertical baffle location on the flow field in rectangular primary sedimentation tanks. Based on the smallest volume of the circulation zone and kinetic energy, the maximum concentration of the suspended sediments in the settling zone and the highest value of removal efficiency, theyprovedthat the baffle (using a baffle height-to-depth ratio of b/H=0.18) should be placed between 0.125 and 0.20 (inlet-to-tank length ratio). Razmi *et al*. (2013) investigated the effect of the baffle position on the performance of a primary settling tank experimentally and numerically. Their results showed that the best position of the bottom baffle (θ=90°) is relatively close to the entrance jet (10-20% tank length), while the best baffle height is around 25-30% of the water depth. The effect of baffle angles and position were examined using a 2D model (Flow-3D 2003) applied to a small-scale, 2-m long laboratory setup (Rostami *et al*. 2011, Shahrokhi *et al*. 2011a, 2012). Right-angled (to the tank base) baffles were most favorable for sedimentation. In addition, it was concluded that, to get high settling performance, the baffle should be somewhere close to the inlet. However the effects of baffle height and optimal baffle configuration were not considered.

The above literature review indicates that most of numerical and experimental studies have been conducted for settling tanks and various inlets. However, less attention is paid to effects angle of baffle on the efficiency of sedimentation basin. Furthermore it should be noted that no research is carried out on irrigation sedimentation basins. The objectives of the present work are to investigate the effect of the bottom baffle and its angle on the performance of an irrigation sedimentation basin experimentally and numerically, and then to studythe effects of Froude number on the sediment removal efficiency.

2. Materials and methods

2.1 The governing 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 general mass continuity equation is

$$\nabla . \vec{V} = 0 \tag{1}$$

In there, $\vec{V}$ is velocity vector. Also, the momentum equations 2D is (Tamayoland Firoozabadi 2006, Tamayol *et al*. 2010, Mehdizadeh and Firoozabadi 2009, Shamloo and Bayat 2008, Tamayol *et al*. 2004)

$$u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x} + \frac{\partial}{\partial x}\left(v\frac{\partial u}{\partial y}\right) + \frac{\partial}{\partial y}\left(v\frac{\partial u}{\partial y}\right) + \frac{\partial}{\partial x}\left(v_t\frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(v_t\frac{\partial u}{\partial y}\right) \tag{2}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left(v\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial y}\left(v\frac{\partial v}{\partial y}\right)+\frac{\partial}{\partial x}\left(v_t\frac{\partial v}{\partial x}\right)+\frac{\partial}{\partial y}\left(v_t\frac{\partial v}{\partial y}\right) \tag{3}$$

In order to close the set of governing equations, turbulence models for calculation of Reynolds stresses are also required. In this paper, the standard *k*-*ε* model is used where two equations for turbulent kinetic energy, *k*, and dissipation rate, *ε*, are

$$\frac{\partial k}{\partial t}+u_i\frac{\partial k}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{\upsilon_t}{\sigma_k}\frac{\partial k}{\partial x_i}\right)+\upsilon_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_j}\right)\frac{\partial u_i}{\partial x_j}-\varepsilon \tag{4}$$

$$\frac{\partial \varepsilon}{\partial t}+u_i\frac{\partial \varepsilon}{\partial x_i}=\frac{\partial}{\partial x_i}\left(\frac{\upsilon_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_i}\right)+C_{1\varepsilon}\frac{\varepsilon}{k}\upsilon_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_j}\right)\frac{\partial u_i}{\partial x_j}-C_{2\varepsilon}\frac{\varepsilon^2}{k} \tag{5}$$

Turbulent viscosity is determined from

$$\upsilon_t=C_\mu\frac{k^2}{\varepsilon} \tag{6}$$

Where the model constants as per Rodi (1980) are

$$C_\mu=0.09,\ C_{1\varepsilon}=1.44,\ C_{2\varepsilon}=1.92,\ \sigma_\varepsilon=1.3,\ \sigma_k=1.0 \tag{7}$$

These values are based on extensive measurement of free air and water turbulent flows, but they can be also used for wall flows. (Rodi 1980) In this work, particle tracking method (PTM) is used for evaluation of the settling tanks performance. PTM is based on an Eulerian-Lagrangian approach. In this method, particles are injected into inlet and they are tracked until they are either escape the through the tank outlet or they are trapped at the tank bottom (Tamayol and Firoozabadi 2006). Efficiency of tank (η) is related to capture efficiency (the number of particles trapped at the bottom: N_t)

$$\eta=\frac{N_i-N_o}{N_i}=\frac{N_t}{N_i} \tag{8}$$

Where N_i is number of injected particles and N_o is the number of escaped particles.

The boundary condition for the inflow (influent) is constant velocity and the effects of the wind and small ripples on flow-field are neglected. In PTM when a particle collides with the bottom surface, it is assumed to be trapped, but for other walls and the free surface it is assumed to be reflected. Standard wall functions are used for turbulence modeling. Second order upwind scheme is adopted for discretization of the convective terms and the pressure-velocity coupling is handled by SIMPLEC method.

2.2 Details of experimental setup

Experiments were carried out at the Fluid Mechanic Laboratory of Kashan Islamic Azad University. The experiments were conducted in a tilting flume having a length of 8 m, a width of 0.30 m, and a depth of 0.50 m with and without baffles. A rectangular sedimentation basin 3.0 m in length and 0.3 m in width was provided at the end of the channel. Fig. 1 shows a view of the

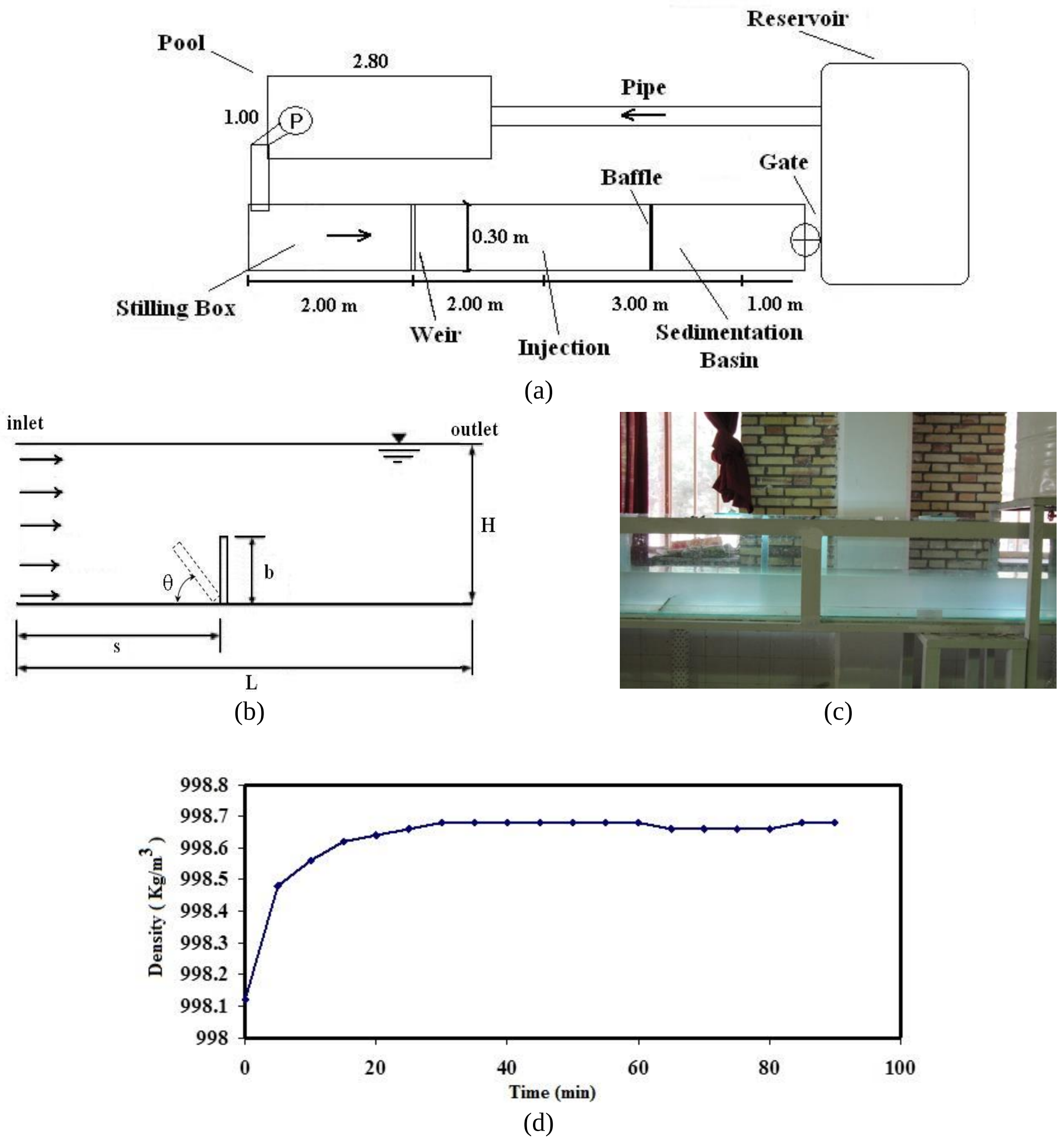


Fig. 1 (a) A view of applied experimental flume; Schematic diagram of the basin; (b) Schematic diagram of the basin; (c) a photo of baffle with θ=60° in the basin; (d) The variations of the density of the outlet flow versus time at the time of injecting the sediments

experimental flume utilized in this study. A sluice gate was provided at the end of the flume for control of the depth of flow (H) within the flume. Water was supplied from a ground pool and total flow rate was measured by a calibrated 60° V notched weir at the beginning of the channel. At the upstream of flume a stilling box has been installed to reduce the kinetic energy of the entrance flow. A pump producing between 1 and 10 L/s replenishes the flume. Fig. 1 shows the sedimentation basin with a length (L) of 3 m, width (W) of 0.3 m and depth of flow (H) of 0.20 m. The height of the baffle was fixed to almost 40% of the total water depth, *b*=0.08 m with different angles (θ). The baffle was located in the middle of the basin (s/L=0.50). Uniform sand with median size of D_{50}=0.130 mm was injected to the sedimentation basin to measure the efficiency of the basin. Natural sand of a relative density of 2.70 was used as the sediment. In this study, out of 63 experiments, 54 were carried out under clear-water conditions.

Froude number (Fr) in reflects the effect of buoyancy forces in the inlet boundary condition and $F_r=U[gH(\rho/\rho_r-1)]^{-1/2}$,where g is the inlet height, U is the inlet velocity, ρ is fluid density and ρ_r is the reference density (clear water, 1000 kg/m^3). Here, the effect of the buoyancy force is quantified using three F_r values of 0.026, 0.063 and 0.116 and six baffle angles (θ) 30°, 45°, 60°, 90°, 120° and 150° and inlet concentration (c) 3 gr/lit in order to investigate the effect of angle baffle on efficiency of the basin. In addition, 9 experiments were conducted under the same condition for the basin with no baffle.

Effects of baffle height and position were not considered. The measurement of efficiency was taken after reaching the equilibrium time. The equilibrium time is when the outlet sediment concentration reaches a constant value. An experiment was conducted for 90 min, with Froude number of 0.026 and inlet sediment concentration 3 gr/lit in basin without a baffle to obtain an equilibrium time. The output fluid density was measured at 5 min intervals for the entire operating period. After analyzing the result, the duration of 30 min for all experimental tests was selected as equilibrium time (Fig. 1(d)). Uniform flow in the channel and sedimentation basin was established by adjusting the sluice gate. The sediment was injected at a constant rate into the flow at the upstream and over the entire width basin. The sediment load downstream of the basin was collected in the reservoir. At the end, the sediment deposited in the sedimentation basin was dried and weighed to determine the efficiency. The removal efficiency of the basin was computed as follows

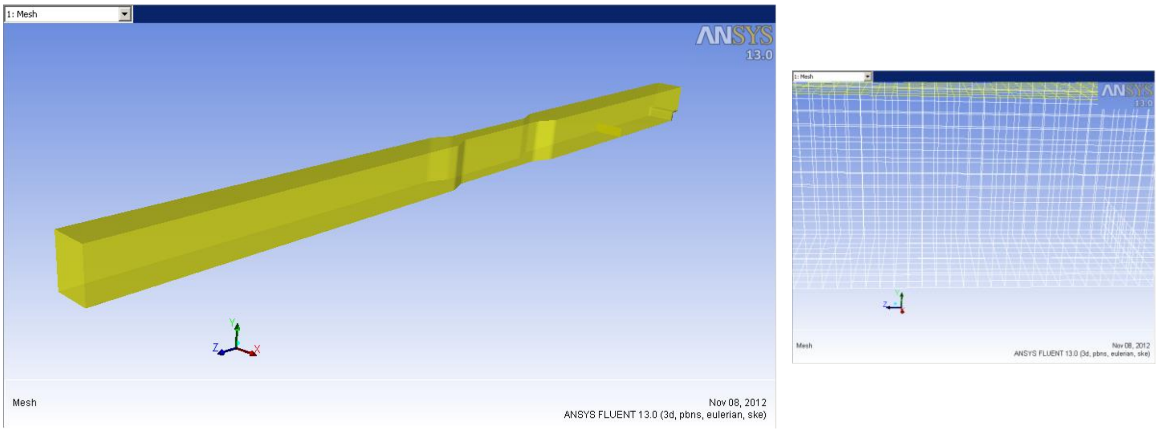
$$\mathrm{RE} = \frac{\mathrm{W_d}}{\mathrm{W_i}} \times 100 \tag{9}$$

Where RE=basin efficiency; and W_i and W_d=Dry weight of sediment entering and depositing in the basin per unit time, respectively. The experiments were repeated for different baffle angles, discharge rates, and inlet sediment concentrations.

2.3 Numerical simulation

Numerical analysis of the basin with baffle was carried out using the CFD package Ansys Fluent. The model was designed in the Gambit tool and exported for meshing in Ansys Fluent. The computational domain considered for the CFD analysis is shown in Fig. 2. Thenumerical flow governing equations are the equation of continuity, Navier-Stokes, equation of momentum and the energy equations for the modeled fluid. The turbulence model is also solved with these equations to calculate the Reynolds stresses.

Due to very low volume fraction of the secondary solid phase that is well below 10% simulations are performed using both the mixture model of the Euler-Lagrange approach of ANSYS FLUENT software package. The Euler-Lagrange approach with Discrete Phase Model (DPM) is used to obtain real distribution of solid phase.DPM is an Euler-Lagrange approach model that tracks individual particles of the secondary phase in a continuous flow of the primary phase. The boundary condition for the inflow (influent) is constant velocity and the effects of the wind and small ripples on flow-field are neglected. When a particle collides with the bottom surface, it is assumed to be trapped, but for other walls and the free surface it is assumed to be



(a) (b)

Fig. 2 Details of computational domain (a) A 3D view of applied numerical modeling of the basin; (b) Meshed geometry of basin

reflected. Standard wall functions are used for turbulence modeling. Second order upwind scheme is adopted for discretization of the convective terms and the pressure-velocity coupling is handled by SIMPLEC method. All simulations were performed via the standard K-ε model while keeping all of the other variables unchanged.

The particle size distribution was defined using the Rosin-Rammler equation based on the assumption that an exponential relationship exists between the particle diameter D, and the mass fraction of the particles with diameter greater than Dithering of the sand particle size was 10-250 μm and the mean diameter and the spread parameter of the Rosin-Rammler distribution function are 130 μm and 4.52, respectively.

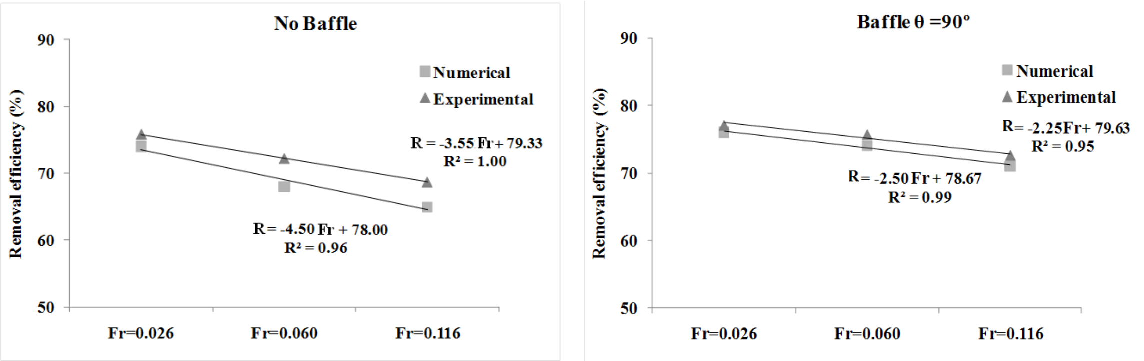
3. Results and discussion

The results and discussion are presented in four parts; the first part discusses the effect of the vertical baffle installed at the bottom of the basin, the second part deals with the angle of the baffle and the last two parts describes the effects of Froude number and flow depth on sediment removal efficiency.

3.1 Effect of the bottom baffle (θ=90°)

Fig. 3 shows the values of efficiency (%) of the basin with and without any baffle for different Froude number. Experimental and numerical Results show that by installing the vertical baffle at the bottom and in the middle of the sedimentation basin, sediment removal efficiency increases by 0.30 to 3.90% compared to the basin without a baffle. Furthermore, the results indicate that with increasing Froude number and concentration, the removal efficiency increases.

A comparison between the removal efficiency for the basin with baffle and no baffle for the same Froude number illustrates that installing the baffle can reduces the velocity near the bed and consequently improves the sedimentation process.



(a) (b)

Fig. 3 The removal efficiency (%) of the basin (a) No baffle; (b) a vertical baffle

Table 1 The removal efficiency (%) of the basin for various cases

Basin	Fr=0.026		Fr=0.060		Fr=0.116	
	Numerical	Experimental	Numerical	Experimental	Numerical	Experimental
No baffle	74.00	75.80	68.00	72.20	65.00	68.70
Baffle (θ=30°)	74.00	76.50	71.00	72.30	67.00	69.10
Baffle (θ=45°)	76.00	77.30	72.00	74.30	69.00	70.80
Baffle (θ=60°)	79.00	78.20	77.00	76.80	73.00	74.00
Baffle (θ=90°)	76.00	77.10	74.00	75.70	71.00	72.60
Baffle (θ=120°)	74.00	76.50	72.00	73.90	68.00	69.70
Baffle (θ=150°)	72.00	74.90	66.00	68.50	61.00	62.80

3.2 Effect of the baffle angle

Table 1 shows the values of the removal efficiency at six baffle angles (θ) 30°, 45°, 60°, 90°, 120° and 150°. The table indicates that the best performance is obtained when the baffle angle θ=60°. The results also show that the difference between the value of efficiency (%) of a sedimentation basin with baffle angle θ=60° increases from 2.5% to 9.0% compared to the basin without any baffle. The maximum value of removal efficiency isin low Froude number.

Streamlines of flow around the baffle near the bed for three angles 60, 90 and 120 degree with the flow are shown in Fig. 4. The dead zones formed downstream the baffle to the direction of flow. The best angle of the baffle is obtained when the volume of the circulation zone is minimized or the dead zone is divided into smaller parts.

The circulation volume is normalized by the total water volume in the tank and calculated by the numerical method in different cases. It is predictable that some cases must have had poor performances. This is related to the size of dead zone. The results shows that the baffle performance with θ=60° is best.

Streamlines of flow around the baffle near the bed for θ=60° with the flow are shown in Fig. 5. For this case, a large vortex in zone downstream is formed. Vortex zones reduce the effective volume of the basins and the effective volume for sedimentation processes, so the suspended particles do not have sufficient space for deposition. A baffle angle of θ=60° indicates that these

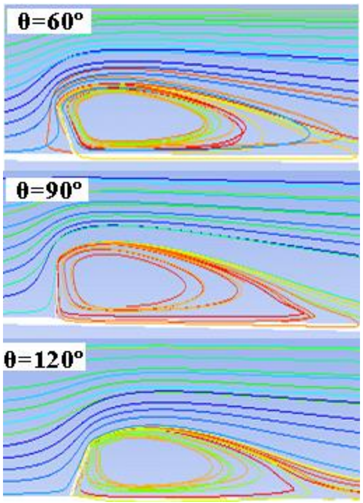
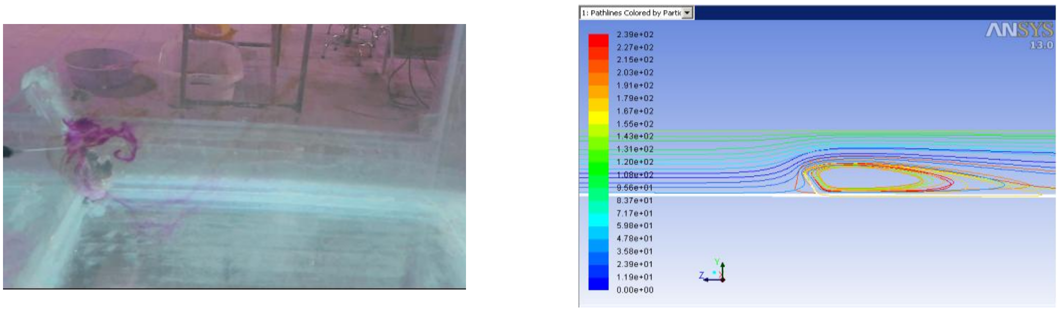
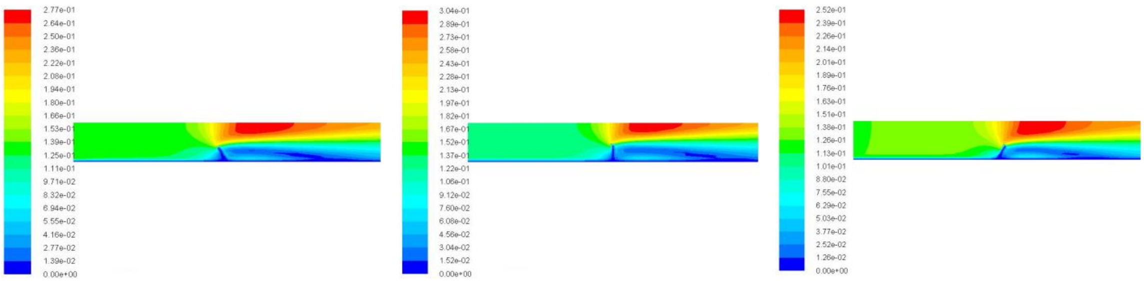


Fig. 4 Effect of angle of a baffle on the flow streamlines



(a) (b)

Fig. 5 Streamlines in the basin with baffle (θ=60°) (a) Experimental; (b) Numerical



θ=60° θ=90° θ=120°

Fig. 6 The contours of velocity (m/s) for different angle (Fr=0.060)

zones are minimized. This means that decreasing the baffle installation angle from 90° to 60° leads to a decrease in the height and the extent of the vortex zones after the baffle.

One of the reasons for using a baffle in basin is reducing the velocity, kinetic energy and reaches to the uniform condition of fluid. Computed contour of velocity for different angles of baffle, (θ=60°, 90° and 120°) are shown in Fig. 6. Comparison between the contours of velocity for

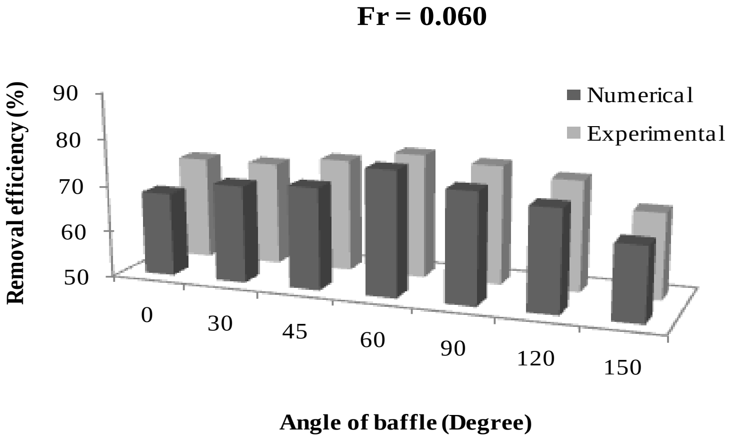


Fig. 7 Variation of efficiency with different angles of baffle for Fr=0.060

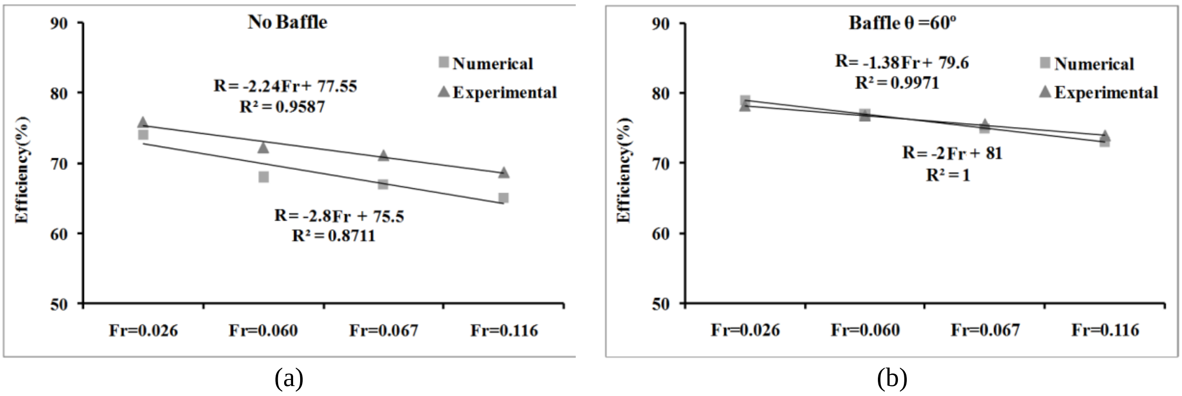


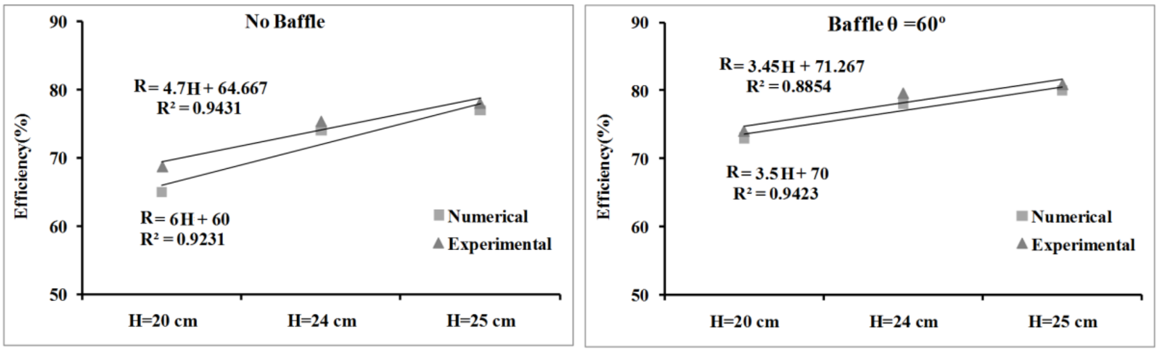
Fig. 8 Removal efficiency of basin for different Froude numbers (a) No baffle; (b) with baffle (θ=60°)

these cases illustrate that baffle (θ=60°) can decrease the length and depth of the maximum magnitude of kinetic energy and velocity and create the better situation for sedimentation process.

The results also indicate that in most experiments, the removal efficiency increases for baffle angles less than 90 degrees, while decreases with baffle angles greater than 90 degrees. The removal efficiency (RE (%)) and baffle angle of baffle (θ) are shown in Fig. 7 for Fr=0.060.

3.3 Effect of Froude number

Four different Froude numbers 0.026, 0.060, 0.067 and 0.116 were applied in order to investigate the effect of flow conditions on the basin efficiency. Fig. 8 shows the effect of Froude number on the values of efficiency (%) for a basin with and without baffle for baffle angle (θ)=60°. Increasing Froude number is associated within increase in the flow velocity and so the removal efficiency decreases. The main reason of such finding is that with increases in Froude number, the vortex expands. Fig. 8 shows that with increasing Fr, the removal efficiency decreases with and without a baffle. The slope of the graph is lower with the baffle angle of 60° compared to without baffle condition. With decreasing Fr, the effect of the baffle on removal efficiency decreases. This



(a) (b)

Fig. 9 Removal efficiency of basin for different flow depth (a) No baffle; (b) with baffle (θ=60º)

is in agreement with the results of Tamayol *et al*. (2010).

3.4 Effect of flow depth

The flow depth (H) was constant for all experiments with a value of 0.20 m. In order to evaluate the effect of flow depth on the removal efficiency of a sedimentation basin with baffle (θ=60º) and without baffle, three flow depths (0.20, 0.24 and 0.25 m) with Fr=0.116 were used.

The results of removal efficiency are shown in Fig. 8. It can be observed that with increasing H, the removal efficiency increases with and without a baffle.

The main reason is that with increasing H and cross sectional area, the velocity decreases. This effect shows itself as an increase in removal efficiency.

The rate of increase of RE with increasing H in a basin with the baffle angle 60° is lower than for a basin without baffle (Fig. 9).

These results indicate that with increasing H, the baffle has less effect on RE. This demonstrates the importance of baffle height.

4. Conclusions

Sedimentation basins are used in irrigation networks for the removal of suspended sediments. Installation of baffles can improve the efficiency of the basin in terms of settling. In this work, the experimental and numerical tests were performed to investigate the effects of baffling and baffle angle on the sediment removal efficiency of an irrigation sedimentation basin. The results show that the installation of a vertical baffle at the bottom and in the middle of the basin improves the efficiency up to 4%, with the baffle height of 40% of the water depth. Alsoat the baffle angle of 60º, the overall removal efficiency increases up to 9.0%. Furthermore, the measured data indicates that by increasing the Froude number and decreasing the depth of flow, the removal efficiencies of the basin with and without baffle are decreased. The numerical and experimental results are found to be in good agreement. The CFD approach presented in this paper will be useful for practicing engineers and it helps in increasing the confidence in the numerical approach for the designers. The DPM model should be used in obtaining accurate distribution of the sediment at the basin.

Also, the Rosin-Rammler equation proposed for particle size distribution in multiphase flows (sediment-water).

References

Ahmed, F.H., Kamel, A. and Abdel Jawad, S. (1996), "Experimental determination of the optimal location and contraction of sedimentation tank baffles", *Water Air Soil Pollut*., **92**, 251-271.

Bretscher, U., Krebs, P. and Hager, W.H. (1992), "Improvement of flow in final settling tanks", *J. Environ. Eng*., **118**(3), 307-321.

Crosby, R.M. (1984), "Evaluation of the hydraulic characteristics of activated sludge secondary clarifiers", Environmental Protection Agency, Office of Research and Development, Washington, D.C. U.S Report EPA-600/2.

Goula, A.M., Kostoglou, M., Karapantsios, T.D. and Zouboulis, A.I. (2007), "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**, 110-121.

Huggins, D.L., Piedrahita, R.H. and Rumsey, T. (2004), "Analysis of sediment transport modeling using computational fluid dynamics (CFD) for aquaculture raceways", *Aquacult. Eng.*, **31**, 277-293.

Krebs, P., Vischer, D. and Gujer, W. (1992), "Improvement of secondary clarifiers efficiency by porous walls", *Water Sci. Technol*., **26**, 5-6.

Krebs, P. (1995), "Success and shortcomings of clarifier modeling", *J. Water Sci. Technol.*, **31**(2), 181-191.

Krebs, P., Vischer, D. and Gujer, W. (1995), "Inlet-structure design for final clarifiers", *J. Environ. Eng*., ASCE, **121**(8), 558-564.

McCorquodale, J.A. and Zhou, S. (1993), "Effects of hydraulic and solids loading on clarifier performance", *J. Hydraul. Res*., **31**(4), 461-477.

Mehdizadeh, A. and Firoozabadi, B. (2009), "Simulation of a density current turbulent flow employing different RANS models, a comparison study", *Tran. B:Mech. Eng.*, **16**(1), 53-63.

RangaRaju, K.G., Kothyari, U.C., Srivastav, S. and Saxena, M. (1999), "Sediment removal efficiency of settling basins", *J. Irrig. Drain. Eng*., **125**(5), 308-314.

Razmi, A.M., Firoozabadi, B. and Ahmadi, G. (2008), "Experimental and numerical approach to enlargement of performance of primary settling tanks", *J. Appl. Fluid. Mech*., **2**(1), 1-13

Razmi, A.M., Bakhtyar, R., Firoozabadi, B. and Barry, D.A. (2013), "Experiments and numerical modeling of baffle configuration effects on the performance of sedimentation tanks", *Can. J. Civil Eng*., **40**(2), 140-150.

Rodi, W. (1980), "Turbulence models and their application in hydraulics", *IAHR*, Delft, the Netherlands.

Rostami, F., Shahrokhi, M., Md Said, M.A., Abdullah, R. and Syafalni, S. (2011), "Numerical modeling on inlet aperture effects on flow pattern in primary settling tanks", *Appl. Math. Model.*, **35**(6), 3012-3020.

Shahrokhi, M., Rostami, F., Md Said, M.A., Sabbagh-Yazdi, S. and Syafalni, S. (2011a), "The effect of number of baffles on the improvement efficiency of primary sedimentation tanks", *Appl. Math. Model.*, **36**(8), 3725-3735.

Shahrokhi, M., Rostami, F., Md Said, M.A., Sabbagh-Yazdi, S., Syafalni, S. and Abdullah, R. (2012), "The effect of baffle angle on primary sedimentation tank efficiency", *Can. J. Civil Eng*., **39**(3), 293-303.

Shahrokhi, M., Rostami, F., Md Said, M.A. and Syafalni, S. (2011b), "Numerical modeling of the effect of the baffle location on the flow field, sediment concentration and efficiency of the rectangular primary sedimentation tanks", *World Appl. Sci. J.*, **15** (9), 1296-1309.

Shahrokhi, M., Rostami, F., Md Said, M.A. and Syafalni, S. (2011c), "Numerical simulation of influence of inlet configuration on flow pattern in primary rectangular sedimentation tanks", *World Appl. Sci. J.*, **15**(7), 1024-1031.

Shamloo, H. and Bayat, A.R. (2008), "Evaluation different turbulent models in hydraulic design of Primary Sedimentation Tank", *4 National Congress of Civil Engineering*, Tehran. (in Persian)

Shetab-Boushehri, S.N., Mousavi, S.F. and Shetab-Boushehri, S.B. (2010), “Design of settling basins in irrigation network using simulation and mathematical programming”, *J. Irrig. Drain. Eng.*, **136**(2), 99-106.

Tamayol, A. (2005), “Effects of baffle configurations on the performance of settling tanks”, MS Thesis, Sharif Univ. of Technology, Iran.

Tamayol, A., Nazari, M., Firoozabadi, B. and Nabovati, A. (2004), “Effects of turbulent models and baffle position on Hydrodynamics of settling tanks”, *Int. Mech. Eng. Con.*, **4**, 12-22.

Tamayol, A. and Firoozabadi, B. (2006), “Effects of turbulent models and baffle position on the hydrodynamics of settling tanks”, *Scientia Iranica*, **13**(3), 255-260.

Tamayol, A., Firoozabadi, B. and Ashjari, M.A. (2010), “Hydrodynamics of secondary settling tanks and increasing their performance using baffles”, *J. Environ. Eng.*, **136**, 32-39.

Vanoni, V.A. (1975), “Sedimentation engineering. ASCE manuals and reports on engineering practice”, *ASCE*, **5**, 582-587.

Vittal, N. and Raghav, M.S. (1997), “Design of single-chamber settling basins”, *J. Hydraul. Eng.*, **123**(5), 469-471.

Wills, R.F. and Davis, C. (1962), “Flow patterns in a rectangular sewage sedimentation tank”, *1st Int Conference on Water Pollution Research*.

Xanthos, S., Gong, M., Ramalingam, K., Fillos, J., Deur, A., Beckmann, K. and McCorquodale, J.A. (2010), “Performance assessment of secondary settling tanks using CFD modeling”, *Water Res. Manage.*, **25**(4), 1169-1182.

Zhou, S., McCorquodale, J. and Vitasovic, Z. (1992), “Influences of density on circular clarifiers with baffles”, *J. Environ. Eng*., ASCE, **118**(6), 829-847.

CC