Al-Azhar University Civil Engineering Research Magazine (CERM)
Vol. (40) No. (2) April, 2018

Modeling the effect of inlet baffle longitudinal and vertical positions on the settling tank performance with computational fluid dynamics

Mostafa Hassanien[1], Emad S. Elmolla[1,2]*, Usama F. Mahmoud[1]

[1]Dept. of Civil Engineering, Faculty of Engineering, Al-Azhar University, Cairo, Egypt.
[2]Adunct Assoc. Prof., Environmental Engineering Programme, Zewail City of Sceince and Technology, 6[th] of October, Egypt.

ملخص:

الترسيب هو عملية أساسية هامة في محطات المياه التقليدية. الهدف من هذا العمل هو دراسة تأثير المكان الافقي والرأسي على خصائص النمذجة لحوض الترسيب و ذلك باستخدام ديناميكية السائل الحسابية (CFD). المسافات الافقية الحاجز المدخل هي 3 و 5 و 7 و 15 % من طول الخزان. بينما المسافات الرأسية 16و 33و 67و 75 % . حدث تقليل كبير لقيمة السرعة في منطقة مدخل الخزان وذلك عند استخدام حاجز المدخل وتم اثبات ذلك باستخدام نمذجة (CFD). افضل مكان راسي كان 33% من عمق الخزان صعودا من قاع الخزان وذلك عند اي موقع افقي للحاجز. افضل مكان افقي 3% من طول الخزان من المدخل.

ABSTRACT

Settling is an important unit operation in conventional water treatment plants. The objective of this work is to study the effect of inlet baffle longitudinal and vertical positions on the hydraulic characteristics of the settling tank modeled using computational fluid dynamics (CFD). The considered inlet baffle longitudinal positions were 3, 5, 7 and 15% of the tank length. However, the considered inlet baffle vertical positions were 16, 33, 67 and 75% vertical bottom contraction of tank depth. Significant reduction in the velocity magnitude at the tank inlet is achieved by the inlet baffle and proved by CFD modeling. The best vertical position was 33% of the tank depth upward from the bottom at any longitudinal position. The best longitudinal position was 3% of the tank length (L) from the inlet.

Keywords: CFD, inlet baffle, settling tank.

1. INTRODUCTION

Problems that may affect the settling tank performance include density current, dead zone, strong surface current, recirculating current, short circuiting, channelling and inefficient sludge removal [1]. When water enters a sedimentation tank, water may not move uniformly from the inlet to the outlet [2]. The short-circuiting may take place due to direct channelling from the inlet to the outlet, wind effects and thermal stratification, [1, 3]. Dead zones are defined as circulation zones in settling tanks and it occupies a considerable volume of the tank, which could decrease the effective volume of settling process [4]. Circulation region or dead zone creates high flow mixing problems in the settling tanks [5]. Thus, decreasing the formation of the dead zones is an important objective of the settling tank design [5]. Suitable baffle configuration is reported to be a suitable methodology for dead zone reduction and influent energy dissipation [6].

Computational fluid dynamics (CFD) calculations have been used to improve the process design [7]. Water flow patterns inside sedimentation tanks may be predicted by solving the partial differential equations using CFD [8]. Applications of CFD modeling for the simulation of the water and wastewater sedimentation tanks have been reported in the literature [9-19]. Razmi et al. [4] used the CFD for verification of the experimental data for optimum baffle location. They concluded that presence of baffle can reduce the turbulent kinetic energy. Shahrokhi et al. [20] studied the effect of baffle location on the flow pattern in a rectangular primary sedimentation tank using experimental work and CFD modeling. The results showed that CFD modeling output agreed with experimental results. Goula et al. [8] studied the use of CFD modeling to evaluate the effect of vertical baffle addition in a full-scale sedimentation tank. They reported that vertical baffle addition decreased the dead zone and enhanced the settling of solids by directing them towards the bottom of the tank. The settling efficiency increased from 90.4% (no-baffle) to 98.6% after baffle addition. Sajjadi et al. [21] applied computational fluid dynamics simulations with FLUENT software to assess the effect of height and position of baffle in irrigation settling basin. Abbas et al. [22] used computational fluid dynamic model to study the performance improvement of water treatment plants. The results showed that the use of baffle force the solids to move faster towards the tank bottom and decrease the inlet recirculation zone. The overall solids removal efficiency increased from 50 to 90.5% after baffle addition. Tamayol et al. [23, 24] studied the effect of a simple baffle at different positions. They concluded that when the baffle is located at improper position or it has improper height, the performance of primary sedimentation tank would be decreased. Adams and Rodi [25] found that that smaller inlet causes less removal efficiency because high velocity due to small inlet may cause recirculation zone.

The objective of this work is to investigate the effect of inlet baffle positions (longitudinal and vertical) on the hydraulic characteristics of the settling tank modeled using computational fluid dynamics (CFD).

2. Materials and methods

In this study the FLUENT computational fluid dynamics (CFD) software was used to model the hydraulic characteristics of settling tank under the different phases of inlet baffle position. Standard k- epsilon model was used to solve the model. The k-ε model is one of the turbulent models that contain two equations. These two equations are the turbulence kinetic energy equation k and the dissipation equation ε. The exact k-ε equations contain many unknown and unmeasurable terms. For more practical approach, the standard k-ε turbulence model was used [16, 26]. The boundary condition for the inlet is the constant velocity. The outlet is indicated as outflow boundary condition. The free surfaces is described as symmetry boundary condition. The tank sizing was 150 cm length, 60 cm, width and water depth 30 cm. The considered inlet baffle longitudinal positions were 3, 5, 7 and 15% of the tank length. However, the considered inlet baffle vertical positions were 16, 33, 67 and 75% of the vertical bottom contraction of tank depth. The flow rate throughout the modeling work was 0.25 l/sec.

3. Results and discussion

Computational Fluid Dynamics (CFD) method is a powerful tool that used to simulate the hydrodynamics and flow behavior in a sedimentation tanks. A two-dimension geometrical model of rectangular settling tank has been developed in design modular associated with ANSYS workbench. The contours of kinetic energy that describe the hydraulic performance of the flow velocities are obtained from the CFD modeling.

The main purpose of the inlet baffle is achieving uniform distribution of the flow across the width of the settling basin and dissipating incoming velocity [1]. A solid movable baffle was used to separate the inlet compartment from the settling basin. Seventeen different longitudinal and vertical positions for the baffle wall were studied at inlet. The positions of the inlet gitudinallonbaffle (that measured (s1from the inlet were3%, 5%, 7% and 15% of the tank length (L). The vertical positions of the inlet baffle (h1) that measured the opening from the tank bottom to the lower edge of the baffle were16%, 33%, 67% and 75% of tank depth. Table 1, summarizes the studied cases for the inlet baffle positions.

Table 1: Studied cases for the inlet baffle positions

Case number	Inlet baffle longitudinal position (s1) as a percentage of tank length measuredfrom the inlet	Inlet baffle position (h1) as a percentage of tank depth upward from the bottom
1	No-baffle	No-baffle (100%)
2	3%	16%
3	3%	33%
4	3%	67%
5	3%	75%
6	5%	16%
7	5%	33%
8	5%	67%
9	5%	75%
10	7%	16%
11	7%	33%
12	7%	67%
13	7%	75%
14	15%	16%
15	15%	33%
16	15%	67%
17	15%	75%

The velocity contours in a no-baffle tank is shown in figure 1. The figure shows high velocity magnitude at the tank surface. The strong surface current generated by the high velocity magnitude could push the particles to flow out of the tank directly without enough time for settling [20]. It also shows high velocity magnitude at the bottom and middle of the tank. The high velocity of the flow at the tank bottom could form re-circulating current that cause re-suspension of the settled particles in the bottom of the tank [27]. Re-circulating current leads to dead zone formation and hence reducing the effective volume of settling tank. Re-circulating current cause mixing which may bring bottom settled particles back to tank surface [28].

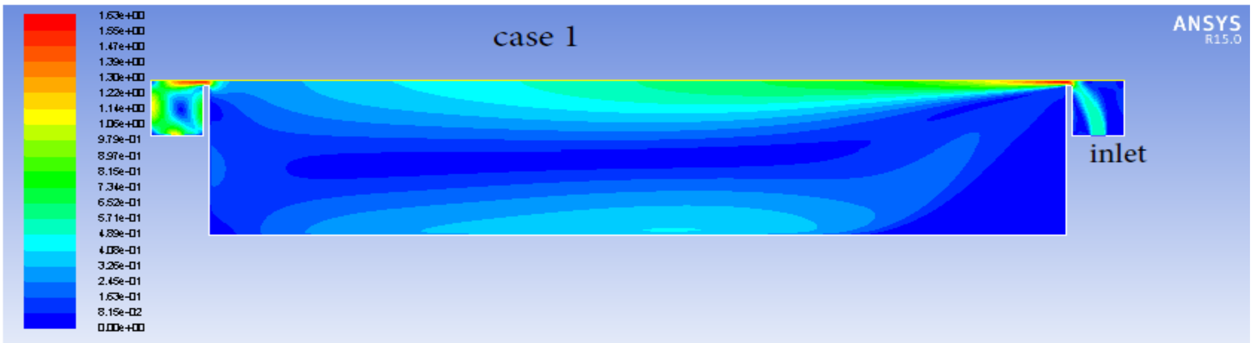

Figure 1: Computed velocity vector contours at no baffles (case 1)

The effect of varying the h1 in the range of 16%, 33%, 67% and 75% at s1 of 3% (Cases 2-5) on the settling tank hydraulic characteristics was studied. Figure 2, shows velocity magnitude for (Cases 2-5), figure showed that the use of inlet baffle at 3% of length significantly improved the settling tank hydraulic characteristics comparing with no inlet baffle. From the figure, the case that has inlet baffle is located 33% of tank depth upward from the bottom appeared to be the optimum vertical position (Case 3). Due to effect of inlet baffle, the high velocity magnitude at the tank surface and at the bottom of the tank is significantly reduced.

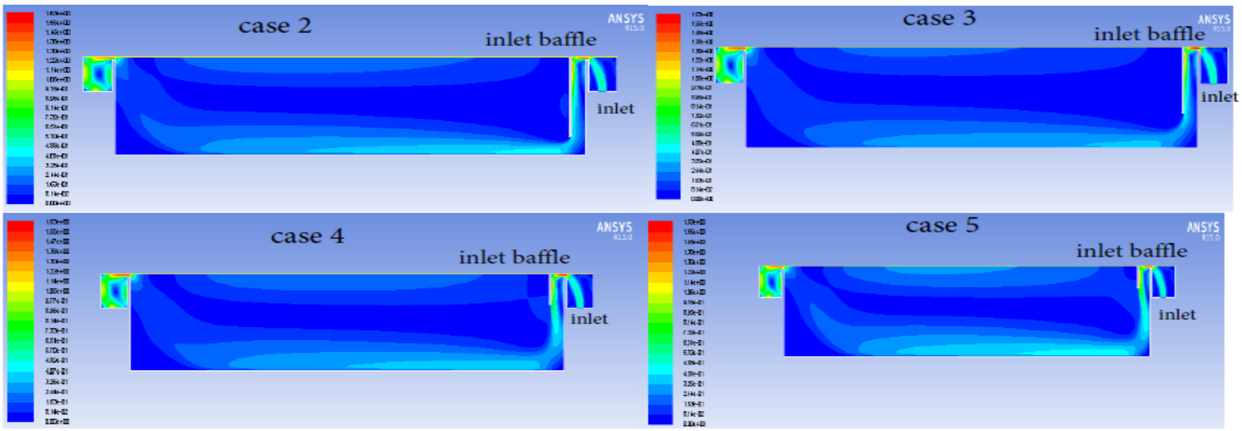

Figure 2: Computed velocity vector contours at inlet baffle (case 2-5)

The inlet baffle position was shifted toward the outlet and s1 was increased from 3% to 5% at h1 of 16%, 33%, 67% and 75% (Cases 6-9). Figure 3, shows velocity magnitude for (Cases 6-9), figure showed the use of inlet baffle at 5% of length which significantly improve the settling tank hydraulic characteristics comparing with no inlet baffle. In comparison with cases6-9 it is noted that best inlet baffle vertical position is depending on the velocity magnitude in the tank. From the figure, the case that has inlet baffle is located 33% of tank depth upward from the bottom appeared to be the optimum vertical position (Case 7). Due to effect of inlet baffle, the high velocity magnitude at the tank surface and at the bottom of the tank is significantly reduced.

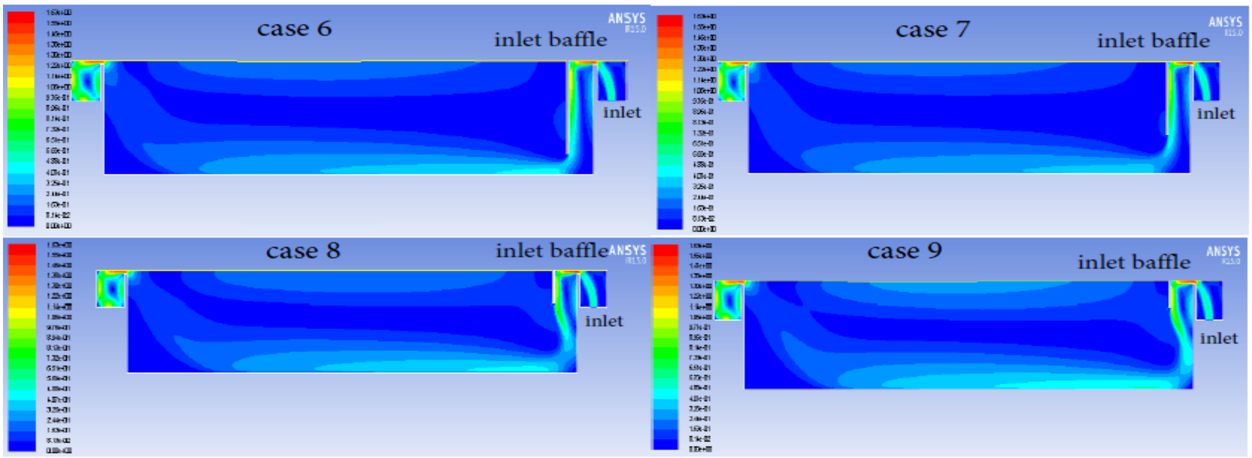

Figure 3: Computed velocity vector contours at inlet baffle (case 6-9)

The inlet baffle position was shifted toward the outlet and s1 was increased from 5% to 7% at h1 of 16%, 33%, 67% and 75% (Cases 10-13). Figure 4, shows velocity magnitude for (Cases 10-13), figure showed that the use of inlet baffle at 7% of length significantly improve the settling tank hydraulic characteristics comparing with no inlet baffle. h casesIn comparison wit 10-13 it is noted that best inlet baffle vertical position is depending on the velocity magnitude in the tank. From the figure, the case that has inlet baffle is located 33% of tank depth upward from the bottom appeared to be the optimum vertical position (Case 11). Due to effect of inlet baffle, the high velocity magnitude at the tank surface and at the bottom of the tank is significantly reduced.

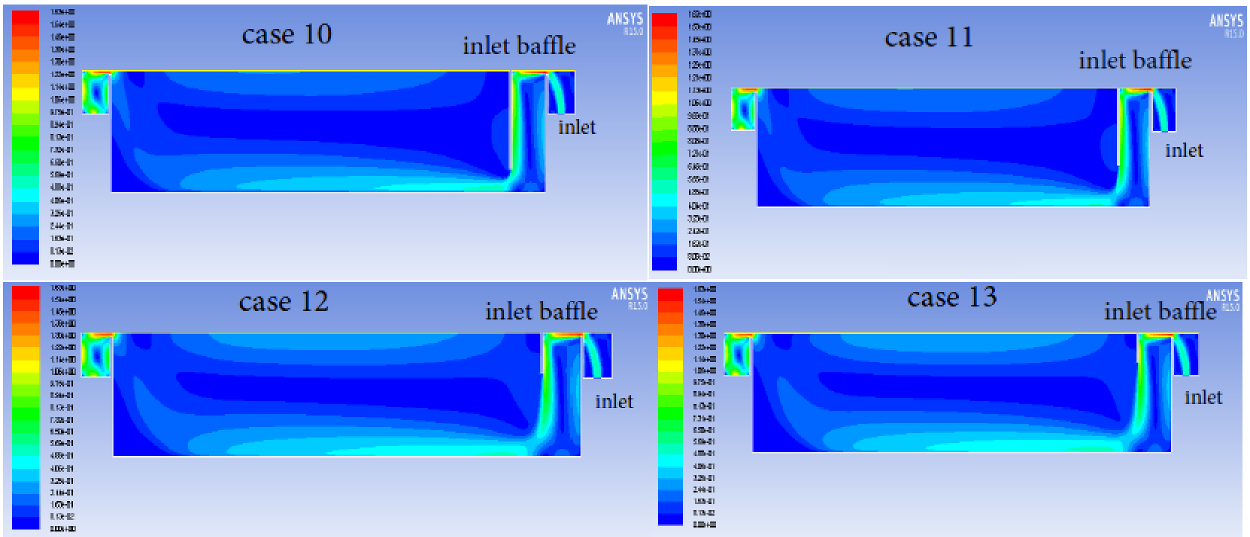

Figure 4: Computed velocity vector contours at inlet baffle (case 10-13)

The inlet baffle position was shifted toward the outlet and s1 was increased from 7% to 15% at h1 of 16%, 33%, 67% and 75% (Cases 14-17). Figure 5, shows velocity magnitude for (Cases 14-17), figure showed that the use of inlet baffle at 15% of length significantly improve the settling tank hydraulic characteristics comparing with no inlet baffle. The high velocity for all cases was at some where before the inlet baffle and at the bottom of the tank but for the tank with no baffle there was high velocity magnitude at whole tank. In comparison with cases 14-17 it is noted that best inlet baffle vertical position is depending on the velocity magnitude in the tank. From figures the case where the inlet baffle is located 33% of tank depth upward from the bottom, appeared to be the optimum vertical position (Case 15). Due to effect of inlet baffle, the high velocity magnitude at the tank surface and at the bottom of the tank is significantly reduced.

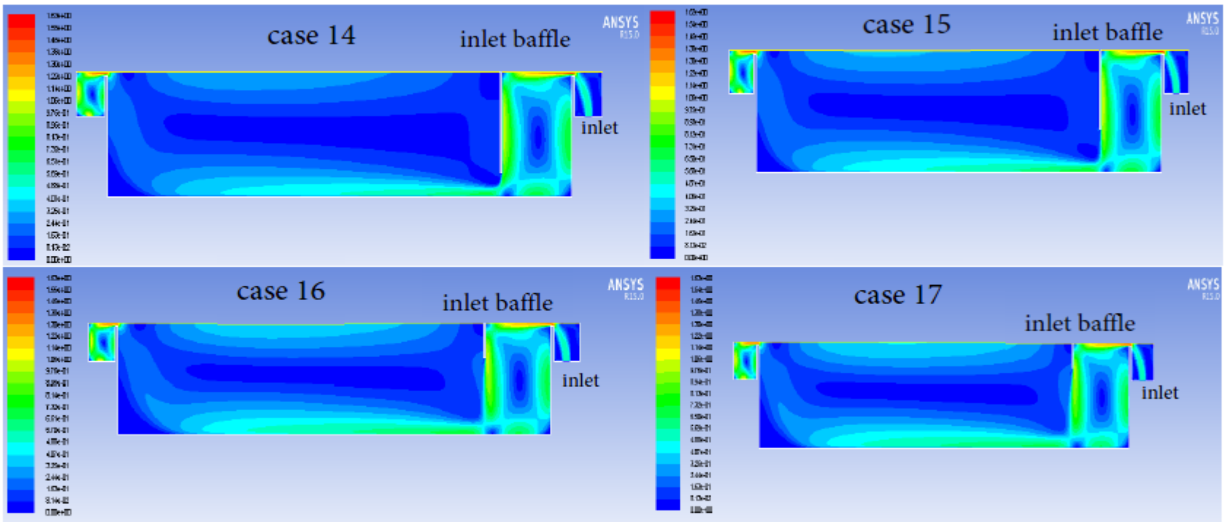

Figure 5: Computed velocity vector contours at inlet baffle (case 14-17)

The inlet baffle best longitudinal and vertical positions determined from the previous cases are compared together as presented in Figure 6. From the figure, the case with inlet baffle is located at 3% of the length and 33% of tank depth upward from the tank bottom appeared to be the optimum inlet baffle position (case 3). The velocity magnitude at the tank surface and at the bottom of the tank is significantly reduced comparing with the other cases.

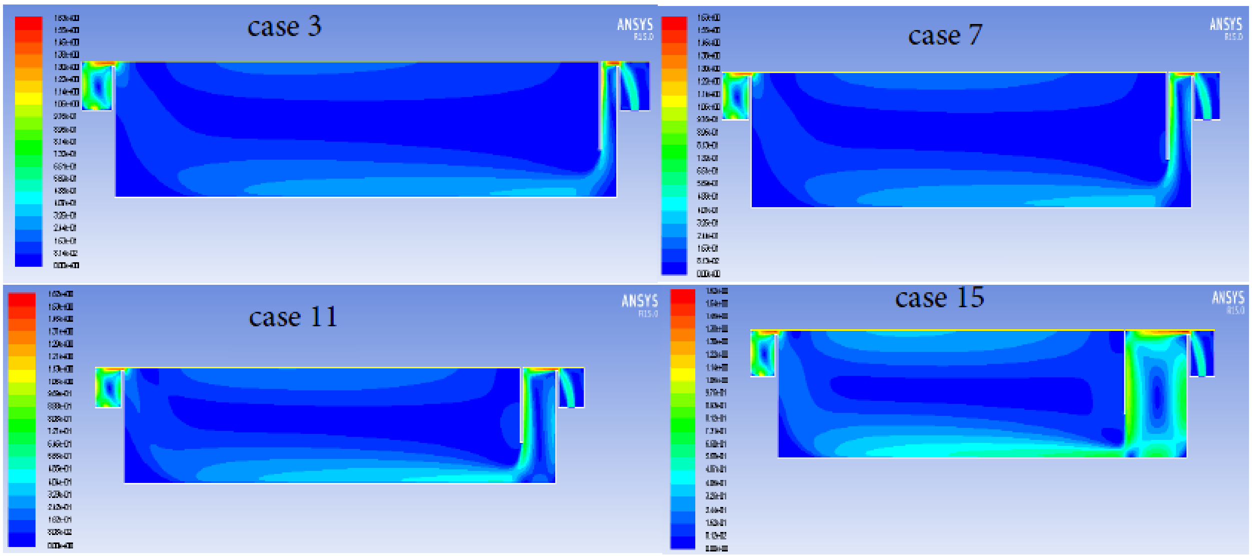

Figure 6: Comparison between different inlet baffle best longitudinal and vertical positions

4. Conclusions

- Significant reduction in the velocity magnitude at the tank inlet is achieved by the inlet baffle and proved by CFD modeling.
- The best vertical position is 33% of the tank depth upward from the bottom at any longitudinal position.
- The best longitudinal position of the baffle is at 3% of the tank length (L) from the inlet.

References

[1] D.G. Stevenson,Water Treatment Unit Process, Imperial College Press, 1997.

[2] A.Tamayol1, B. Firoozabadi1 and G. Ahmadi, Determination of Settling Tanks Performance Using an Eulerian-Lagrangian Method, Journal of Applied Fluid Mechanics, Vol. 1, No. 1, pp. 43-54, 2008.

[3] J. Oca, Ingrid Masaló, Lourdes Reig, Comparative analysis of flow patterns in aquaculture rectangular tanks with different water inlet characteristics, Aquacultural Engineering 31 (2004) 221–236.

[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) 13.

[5] M. Shahrokhi, F. Rostami, M. A. M. Said, S.R. S. Yazdi, Syafalni, The effect of number of baffles on the improvement efficiency of primary sedimentation tanks, Applied Mathematical Modelling 36 (2012) 3725–3735.

[6] P. Krebs, D. Vischer, W. Gujer, Inlet-structure design for final clarifiers, Journal of Environmental Engineering, ASCE 121 (1995) 558–564.

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

[8] Goula, A.M., Kostoglou, M., Karapantsios, T.D., Zouboulis, A.I., CFD methodology for the design of sedimentation tanks in potable water treatment Case study: The influence of a feed flow control baffle, Chemical Engineering Journal 140 (2008) 110–121.

[9] Shamber, D.R., Larock, B.E., Numerical analysis of flow in sedimentation basins, J. Hydr. Div. ASCE 107 (HY5) (1981) 575–591.

[10] McCorquodale, J.A., Yuen, E.M., Vitasovic, Z., Samstag, R., Numerical simulation of unsteady conditions in clarifiers, Water Poll. Res. J. Can. 26 (1991) 201–222.

[11] McCorquodale, J.A., Zhou, S., Effects of hydraulic and solids loading on clarifier performance, J. Hydr. Res. 31 (1993) 461–477.

[12] Zhou, S., McCorquodale, J.A., Godo, A.M., Short circuiting and density interface in primary clarifiers, J. Hydr. Eng. 120 (1994) 1060–1080.

[13] Krebs, P., Vischer, D., Gujer, W., Inlet-structure design for final clarifiers, J. Environ. Eng. 121 (1995) 558–564.

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

[15] Imam, E., McCorquodale, J.A., Bewtra, J.K., Numerical modeling of sedimentation tanks, J. Hydr. Eng. 109 (1983) 1740–1754.

[16] Stamou, A.I., Adams, E.A., Rodi, W., Numerical modelling of flow and settling in primary rectangular clarifiers, J. Hydr. Res. 27 (1989) 665– 682.

[17] Adams, E.W., Rodi, W., Modelling flow and mixing in sedimentation tanks, J. Hydr. Eng. 116 (1990) 895–913.
[18] Lyn, D.A., Stamou, A., Rodi, W., Density currents and shear induced flocculation in sedimentation tanks, J. Hydr. Eng. 118 (1992) 849–867.
[19] Frey, P.H., Hydrodynamics fields and solid particle transport in a settling tank, J. Hydr. Res. 31 (1993) 763–776.
[20] Shahrokhi, M., Rostami, F., Said, M.A.M., Syafalni, Numerical modeling of baffle location effects on the flow pattern of primary sedimentation tanks, Applied Mathematical Modelling 37 (2013) 4486–4496.
[21] Sajjadi, S.M., M. Shafai Bejestan and M. Bina, 2005. Effect of baffle in irrigation settling basin with CFD. 3th national water resources water of Iran, Tabriz (In Persian).
[22] Abbas A. Al-Jeebory, Josef Kris, Ali H. Ghawi. PERFORMANCE IMPROVEMENT OF WATER TREATMENT PLANTS IN IRAQ BY CFD MODEL. Al-Qadisiya Journal For Engineering Sciences, 2010, Vol. 3 No. 1
[23] Tamayol, A. and B. Firoozabadi, 2006. Effects of turbulent models and baffle position on hydrodynamics of settling tanks, Scientia Iranica J., 13(3): 255-260.
[24] Tamayol, A., B. Firoozabadi and G. Ahmadi, 2006. Increasing performance of final settling tanks by using baffles. 7th International Conference on Hydroinformatics, HIC, Nice, France.
[25] Adams, E.W. and W. Rodi (1990). Modeling flow and mixing in sedimentation tanks. J. of Hydraulic Engineering 116 (7), 895-913.
[26] Launder, B.E.; Spalding, D.B. (March 1974). "The numerical computation of turbulent flows". Computer Methods in Applied Mechanics and Engineering. 3 (2): 269–289.
[27] Tarpagkou, R., Pantokratoras, A., CFD methodology for sedimentation tanks: The effect of secondary phase on fluid phase using DPM coupled calculations, Applied Mathematical Modelling 37 (2013) 3478–3494.
[28] H. Guo, S. J. Ki, S. Oh, Y. M. Kim, S. Wang, J. H. Kim, Numerical simulation of separation process for enhancing fine particle removal in tertiary sedimentation tank mounting adjustable baffle, Chemical EngineeringScience158(2017)21–29.