ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

IJESRT
INTERNATIONAL JOURNAL OF ENGINEERING SCIENCES & RESEARCH TECHNOLOGY

Numerical Model for Water Flow and Settling Behavior in the Sedimentation Basin of Al-Karamah Treatment Plant

Ali Salim Joodi*, Mohammed Ali Rashid Hameed
Environmental Engineering Department , College of Engineering , Al-Mustansiriya University, Baghdad, Iraq

Abstracts

The objective of the present work is to develop a two dimensional numerical model to predict the performance of the sedimentation basin in Al-Karamah treatment plant and consequently to increase the basin efficiency. The mixture model adopted by COMSOL Multiphysics is applied coupling with k-epsilon turbulent model. The model showed that particles commence out from the basin after 5 min and it commence to collect at the sludge zone after 1.6 hr. The model demonstrated the important role of SOR on the performance of the sedimentation basin. Also, the result showed that the removal efficiency increases with a decrease of SOR. The relationship between SOR and removal efficiency is nonlinear second order equation with R^2 equal to 0.999. Finally, numerical results demonstrated that the effect of the installation of conical reaction wall on the settling behavior is very small and negligible.

Keywords:

Introduction

Sedimentation basins are the workhorses of any water purification process. It is thus crucial for the sedimentation basin to be operated to its full potential. Generally, the ability of sedimentation basins to clarify water by letting suspended solids settle down as flocculated particles depends on two aspects: (a) the water flow pattern through the basin, which in turn is determined by the configuration of the basin and by operational parameters (solids concentration, water flow rate and temperature) and (b) the settling characteristics of the particles as determined by their shape, size and interaction with the water through drag and buoyancy forces[1,2,3]. The determination of the removal efficiency of a sedimentation basin has been the subject of numerous theoretical and experimental studies[1]. In this context, numerical modeling is very important to describe the flow pattern and solids removal in the sedimentation basin. Finite volume method is used by Shamber and Larock [4] to solve the Navier–Stokes equations, thek –ε model and a solids concentration equation with a settling velocity to model secondary clarifiers.McCorquodale et al. [5] developed a model using a combination of finite element methods (for the stream function) and finite difference methods (for the boundaries). McCorquodale and Zhou [6] investigated the effect of various solids and hydraulic loads on circular clarifier. With respect to primary sedimentation basin, where the solids concentration is limited and discrete settling prevails. E.Imam et al. [7] applied a fixed settling velocity and used an averaged particle velocity. Stamou et al. [8] simulated the flow in a primary sedimentation basin using a 2D model in which the momentum and solid concentration equations were solved but not linkedto account for buoyancy. Adams and Rodi [9] used the same model and did extensive investigations on the inlet arrangements and the flow through curves. More advanced is the work of Lyn et al. [10] that accounts for flocculation where six different size classes with their respective velocities were considered. Frey [11] used the VEST code to determine the flow pattern in a sedimentation basin. The flow profiles were then used by the TRAPS code to determine particle tracks. Van der Walt [12] used the 3D Flo++ code to determine the sensitivity of a primary sedimentation basin behavior on a number of geometric, fluid and solids transport properties and simulated the existing Vaal kop sedimentation basins using a 3D pseudo twophase model demonstrating howthe inlet geometrywas the main cause of the poor desludging capacity.

Generally, Most previous researchesdepends onComputational fluid dynamic (CFD) simulations as a tool to describe water flow and solids removal in sedimentationbasins for water treatment. However, works in COMSOL Multiphysicsmodeling of sedimentation basins for potable water treatment have not been found in the literature. Moreover, the physical characteristics of the flocs may not be such significant parameters in the flow field of clarifiers for potable water, due to the much lower solids concentrations and greater particle size distributions than those encountered

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

in wastewater treatment. The first objective of the present work is to develop a two dimensional mathematical model using COMSOL Multiphysics to simulate the flow and settling behavior of particles in the sedimentation basin of Al-Karamahwater treatment plant. Second objective is to investigate parameters that effect on the flow and settling behavior within thebasin and consequently to improve the performance of the sedimentation basin in Al-Karamahwater treatment plant.

Material and method

Al-Karamahwater treatment plant receives raw water from Tigris River in Baghdad city. It consists from flash mixer, coagulation unit, flocculation unit, sedimentationbasins, filtration unit, and chlorination. The maximum capacity of the plant is 94630 m^3/day. In this plant, there are two circular sedimentation basins with equal dimension. Each one received 2170 m^3/h of raw water with a diameter of 38m.The velocity enters to the basin is 0.76 m/s and detention time equal to 2.1 hr. In the middle of the basin, a reaction well is existed as shown in Fig 1. This well may be to increase the turbulent behavior in the basin and consequently to increase the efficiency of particles removal.

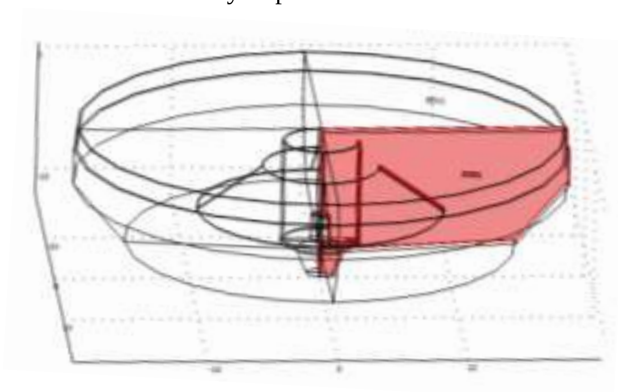

Fig. 1 Circular sedimentation basin in Al-Karamahwater treatment plant

Model development

Traditionally, the flow and settling behaviors in a sedimentation basin are simulated by using NavierStockes equation, k-epsilon turbulence model and advection dispersion equation, separately. In the present work the flow in the sedimentation basinwill be computed with the mixture model that is part of the Chemical Engineering module of COMSOL Multiphysics. The mixture model is able to compute the flow for a mixture of two liquids or a liquid and a solid. The model combines the k-epsilon turbulence model for the main flow with equations for the transport of the dispersed phase and the relative velocity of both phases. Because of the torus-shape of the basin, the model is applied in a 2D geometry with axial symmetry.The mixture model uses the following equations:

$$\rho\frac{\partial u}{\partial t}+\rho(u.\nabla)u=-\nabla p-\nabla.(\rho c_d)u_{slip}u_{slip}+\nabla.\tau_{Gm}+\rho g \quad (1)$$

$$(\rho_w-\rho_s)\left[\nabla.\left(\emptyset_s(1-c_d)u_{slip}-D_{md}\nabla\emptyset_s\right)\right]+\rho_w(\nabla.u)=0 \quad (2)$$

$$\frac{\partial}{\partial t}(\emptyset_s\rho_s)+\nabla.(\emptyset_s\rho_s u_s)=0 \quad (3)$$

Where$u$ denotes mixture velocity (m/s), ρ mixture density (kg/m^3), p pressure (Pa), c_d mass fraction of the solid phase (kg/kg). Furthermore, u_{slip} is the relative velocity between the two phases (m/s), τ_{Gm} the sum of viscous and turbulent stress (kg/m·s^2), and g the gravity vector (m/s^2), ρ_s and ρ_w the density of the solid and water, respectively. The mixture velocity (m/s) is defined as:

$$u=\frac{\emptyset_w\rho_w u_w+\emptyset_s\rho_s u_s}{\rho} \quad (4)$$

Where $\emptyset_w$ and$\emptyset_s$ denote the volume fractions of the water (continuous) phase and the solid (dispersed) phase (m^3/m^3), Where u_w andu_s denote the velocity of the water (continuous) phase and the solid (dispersed) phase (m/s). Therelation between the velocities of the two phases is defined by:

$$u_s-u_w=u_{slip}-\frac{D_{md}}{(1-c_d)\emptyset_s}\nabla\emptyset_s \quad (5)$$

The slip velocity and mixture density can be calculated according to:

$$u_{slip}=-\frac{(\rho-\rho_s)d_s^2}{18\rho\eta}\nabla p \quad (6)$$

$$\rho=\emptyset_w\rho_w+\emptyset_s\rho_s (7)$$

The solid dispersion coefficient is calculated by:

$$D_{md}=\frac{\eta_T}{\rho\sigma_T} \quad (8)$$

Where η_T is the turbulent viscosity (Pa·s) and σ_T is the (dimensionless) turbulent Schmidt number.The turbulence kinetic energy (k) is found by solving:

$$\rho\frac{\partial k}{\partial t}-\nabla.\left[\left(\mu+\rho\frac{C_\mu}{\sigma_k}\frac{k^2}{\varepsilon}\right)\nabla k\right]+\rho U.\nabla k=0.5\rho C_\mu\frac{k^2}{\varepsilon}(\nabla U)^2-\rho\varepsilon \quad (9)$$

And the dissipation rate of turbulent kinetic energy (ε)by solving:

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

$$\rho \frac{\partial \varepsilon}{\partial t} - \nabla . \left[\left(\mu + \rho \frac{C_\mu}{\sigma_\varepsilon} \frac{k^2}{\varepsilon} \right) \nabla \varepsilon \right] + \rho U . \nabla \varepsilon = 0.5 \rho C_{\varepsilon 1} C_\mu k (\nabla U)^2 - \rho C_{\varepsilon 2} \frac{\varepsilon^2}{k} \quad (10)$$

Where k refers to the turbulence kinetic energy (m2/s2), ε is the dissipation rate of turbulent kinetic energy (m2/s3), and C_μ and σ_k are model constants.

The model constants in the previous equations are experimentally determined. Set them to the values listed in Table (1).

Table (1) Model constants[13,14,15]

Constant	C_μ	$C_{\varepsilon 1}$	$C_{\varepsilon 2}$	σ_k	σ_ε
Value	0.09	1.44	1.92	1.0	1.3

Boundary conditions and numerical method

The model describe the water flow and settling behavior in the sedimentation basin usingCOMSOL Multiphysics as a tool to solve partial differential equations. Because of the torus shape of the basin, the model is applied in a two dimension geometry with axial symmetry.Figure 2 shows the cross section of the half sedimentation basin in Al-Karamahwater treatment plant. The bottom has a slight slope towards the center of the basin. The bottom floors have a steep slope of 2.5◦. The basin is center-fed via a pipe located in the bottom of the basin, which has a diameter of 1m. In this context, the water moves from the bottom to the top of the basin and then drops on the reaction well to finally distributed along the basin.The model area is divided into the triangles pane mesh in each aquifer in which the total numbers of elements and mesh points are 2196 and 1212, respectively.The inlet was specified as a plug flow of water at 0.76 m/s, whereas the inlet turbulence intensity was set at 5%. The outlet was specified as a constant pressure outlet. The mass flow rate inlet to the basin is 0.025 kg/s using a measured solid concentration of 20 mg/l, whereas the particle density is 1062 kg/m^3. The water surface is modeled as symmetric plane where the vertical velocity and normal gradient of velocity and turbulent kinetic energy equal to zero.The mixture model for the basin is solved via a transient simulation.

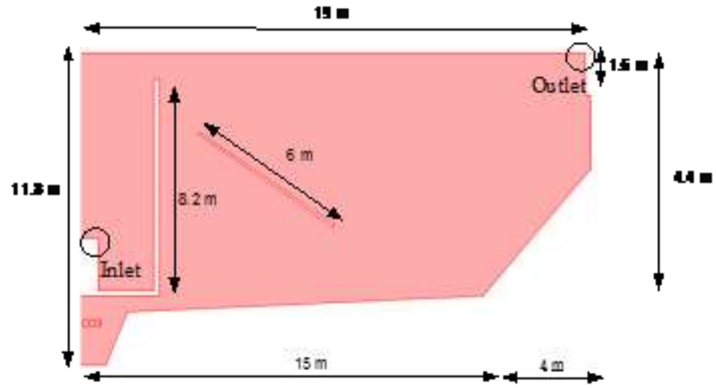

Fig. 2 Cross section of the half sedimentation basin

Results and discussion

The flow and settling behaviors in a sedimentation basin are affected by the basin geometry, surface overflow rate, hydraulic detention time and particles characteristics. In the sedimentation basin of Al-Karamah treatment plant, the surface overflow rate is 1.91 m/hr, hydraulic detention time is 2.1 hr and the mass flow rate inlet to the basin is 0.025 kg/sec, consequently, the actual efficiency of the basin is 62%. In this context, fig 3 presents the velocity streamline and the settling behavior in the basin (color scale) ofAl-Karamahtreatment plant.Generally, the flow pattern is characterized by a large recirculation region spanning a large part of the basin from top to bottom. The predicted streamline refers to the more recirculation eddy around the conical wall and this recirculation decreases reaching to the outlet of the basin. Also, the predicted streamline appears that the dead zone is not existed in the basin, may be as a result to the right side wall. Concerning to the settling behavior, it can be clearly observed that more concentration of particles move toward the bottom of the basin and consequently to the collection zone. Finally, it can be said that the hydraulic detention time is very good.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

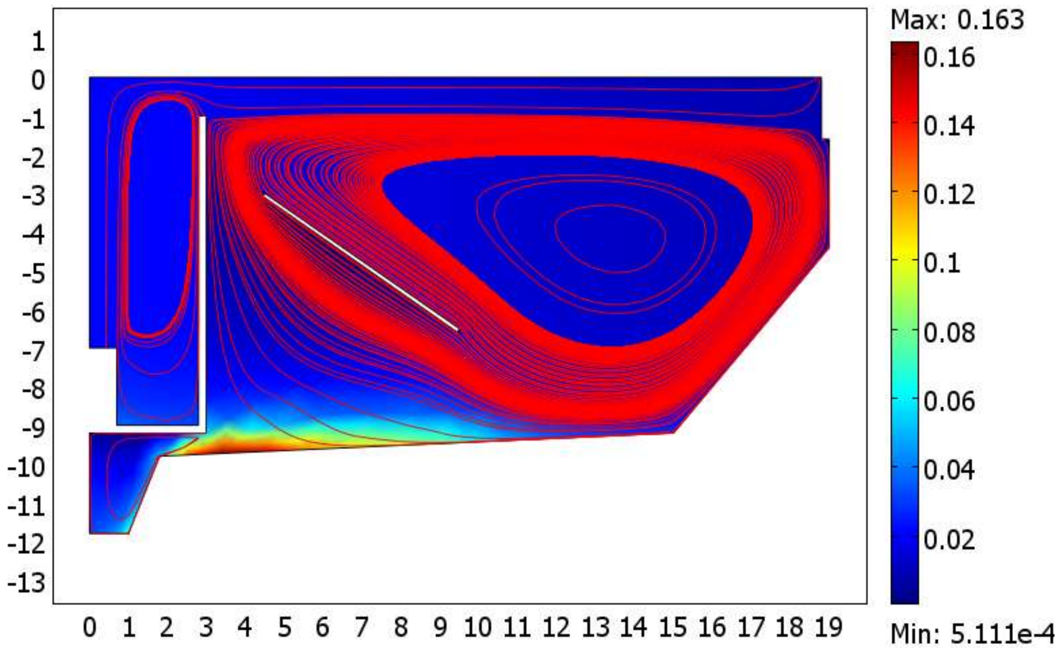

Fig.3 Velocity streamline and settling behavior along the sedimentation basin

Figure 4 illustrates the settling behavior as a function of time. It can be shown that initially the basin contain the pure water. On the other hand, when the time increase, the particles is distributed along the basin and it commences to settle in the incline bottom wall. From this figure, it can be observed the effect of hydraulic detention time on the settling behavior along the basin.

Figure 5 shows the particles concentrations at three points, namely close to the inlet zone, outlet zone and sludge zone. It can be clearly found that the particles concentration is about two times lower at the outlet zone compared with that at inlet zone. Particles commence to out from the basin after 5 min. This can be attributed to the large value of the discharge flow rate input to the basin. Furthermore, particles commence to collect at the sludge zone after 1.6 hr. This may be due to the geometry of the basin or to the effect of conical reaction wall.

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

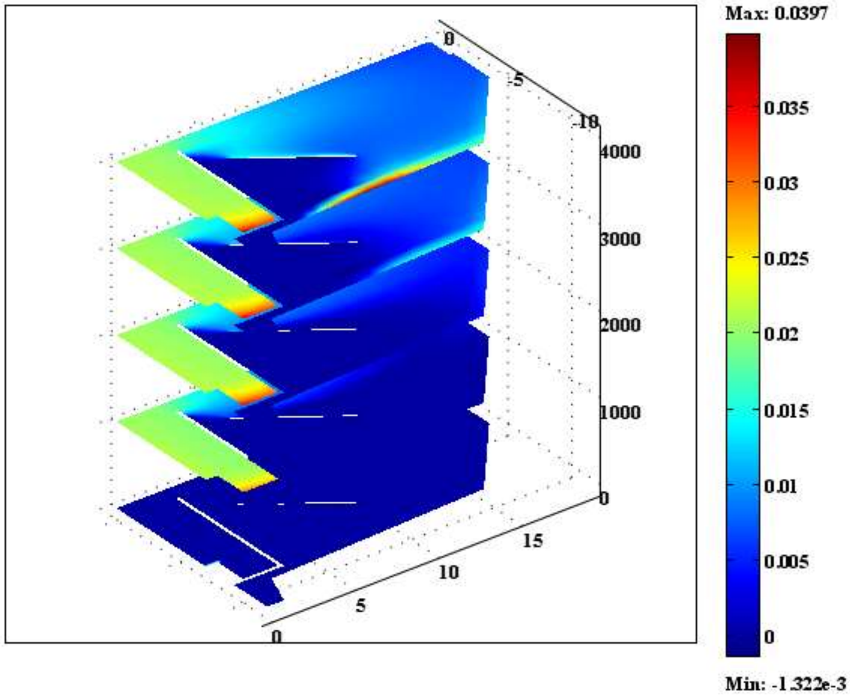

Fig.4 Flow and settling behaviors as a function of time

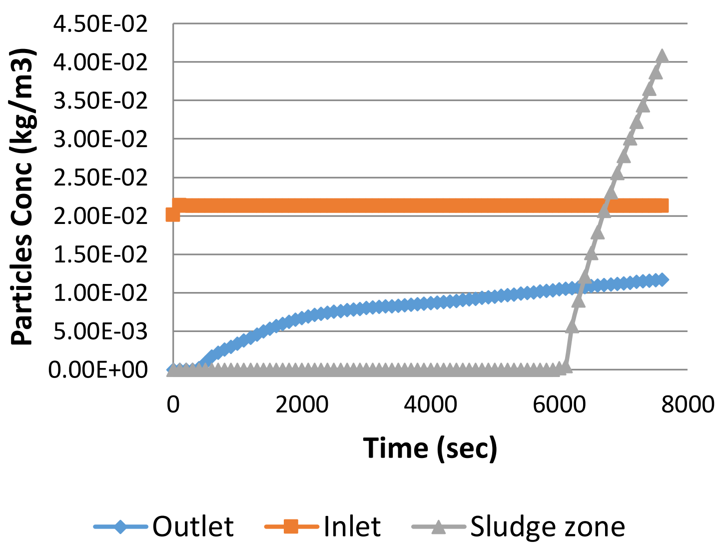

Fig.5Comparison of the particle concentration at the inlet, outlet and sludge zones

Surface overflow rate is defined as the mean parameter effects on the flow and settling behaviors. Generally, when the water contained colloidal particles enters the basin, the basin will be closed from 2 hr to 4 hr and then the water start to move up, this velocity is called SOR. Then SOR equal to the flow rate divided on the surface area. When the surface overflow rate increases this means increase in the water flow rate and by consequence increase of turbulent in the basin. To test the effect of SOR on the efficiency of the sedimentation basin four values is provided, as shown in fig 6 a and b.Figure6a shows the particles concentration at the point close to the outlet zone. Any increase in the value of SOR leads to an increase inthe turbulent behavior in the basin and consequently decreasing in the settling particles. Then it can be concluded that the efficiency of the basin decrease with an increase in the value of SOR. In this context, figure 6b shows the relationship between the removal efficiency of the sedimentation basin with

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

SOR. It can be shown that the efficiency of the basin decreases with any increase in the value of SOR, and the relation between them is nonlinear second order type equation with R^2 equal to 0.999.

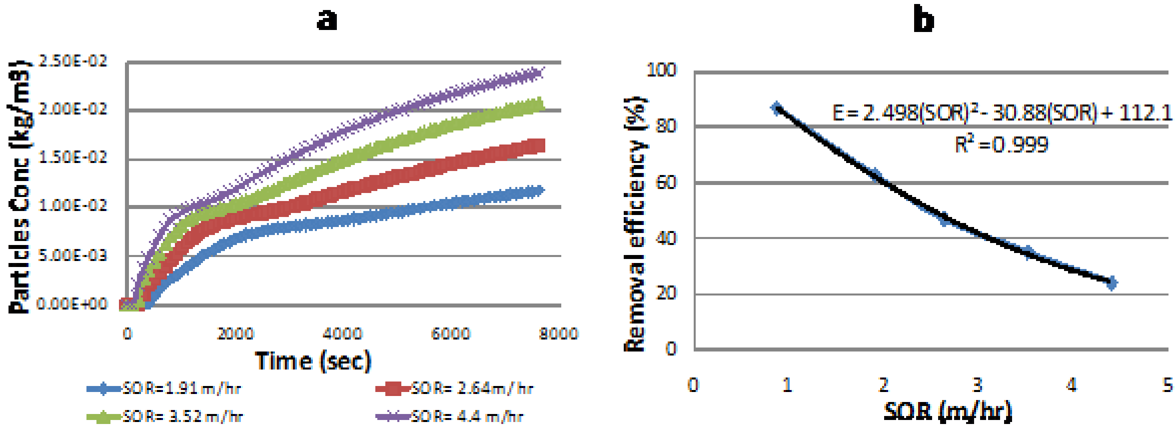

Fig. 6(a) Effect of SOR on the particles concentration at the outlet of the sedimentation basin
(b) Relationship between SOR and removal efficiency of the sedimentation basin

The effect of conical reaction wall installation in the sedimentation basin is presented in fig 7. It can be found that the streamline with the presence of the conical reaction wall refers to the more recirculation eddy around the conical wall and this recirculation decreases reaching to the outlet of the basin. On the other hand, the recirculation eddy decreases at the absence of conical reaction wall. For two cases, the predicted streamline appears that the dead zone is not existed in the basin.

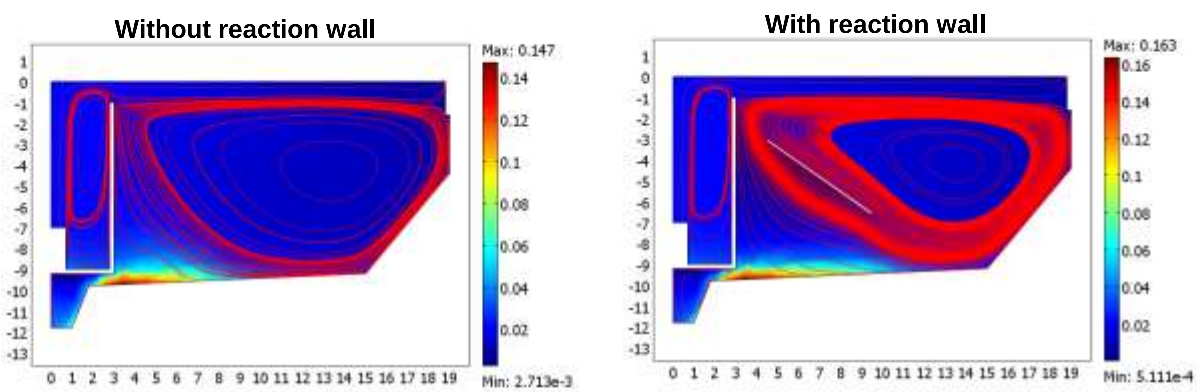

Fig.7 Effect of conical reaction wall on the flow and settling behaviors along the basin

The effect of the reaction wall on the removal efficiency of the sedimentation basin is investigated, as shown in fig 8. It can be clearly observed that particles concentrations at the outlet zone of the basin is not affected by the presence or absence of the conical reaction wall. May be, the influence of the reaction wall is appeared with high value of solid mass flow rate.

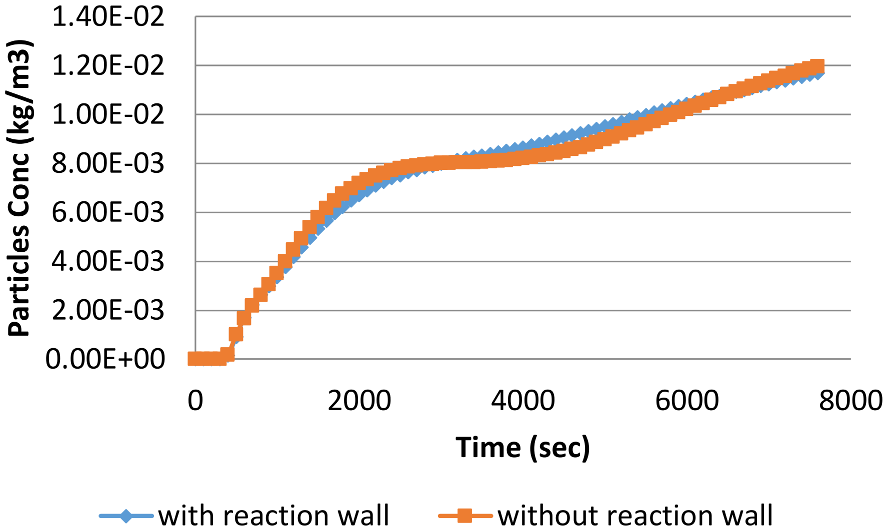

Fig.8 Effect of conical reaction wall on the particles concentration at the outlet of the sedimentation basin

Conclusion

A two dimensional numerical model is developed to investigate the performance of a sedimentation basin in Al-KARAMAH treatment plant. The mixture model that is part of the Chemical Engineering module of COMSOL Multiphysics is applied in the present work. It can be concluded that the mixture model adopted from COMSOL Multiphysics can be used o evaluate the flow and settling behaviors in the sedimentation basin. According to the basin geometry and real data supplied by Al-KARAMAH treatment plant, the results showed that the flow pattern is characterized by a large recirculation eddy around the conical wall. Also, the predicted streamline appears that the dead zone is not existed in the basin. The effect of SOR on the performance of sedimentation basin is investigated in the present work. Any increase in the value of SOR leads to a decrease in the settling particles as a result to the turbulent flow which is increased with increasing of SOR. The results demonstrated that the relationship between SOR and removal efficiency is nonlinear second order equation with R^2 equal to 0.999. Finally, it can be concluded that the installation of conical reaction wall is not effected on the flow and settling behaviors in the sedimentation basin.

References

1. Adams E.W., &Rodi W., (1990). Modelling flow and mixing in sedimentation tanks, J. Hydr. Eng. 116 895–913.
2. Deininger A., Holthausen E., &Wilderer P.A., (1998).Velocity and solids distribution in circular secondary clarifiers: full scale measurements and numerical modelling, Water Res. 32 2951–2958.
3. Ghawi, A.H., (2008). A numerical model of flow and settling in sedimentation tanks in potable water treatment plants, Edíciavedeckýchprác (Science Works Edition) ,Workbook no. 64, STU Bratislava, pp. 150, ISBN 978-80-227-2964-2.
4. D.R. Shamber, B.E. Larock.,(1981). Numerical analysis of flow in sedimentation basins, J. Hydr. Div. ASCE 107 (HY5) 575–591.
5. J.A. McCorquodale, E.M. Yuen, Z. Vitasovic, R. Samstag.,(1991). Numerical simulation of unsteady conditions in clarifiers, Water Poll. Res. J. Can. 26 201–222.
6. J.A. McCorquodale, S. Zhou., (1993).Effects of hydraulic and solids loading on clarifier performance, J. Hydr. Res. 31 461–477.
7. E. Imam, J.A. McCorquodale, J.K. Bewtra.,(1983). Numerical modeling of sedimentation tanks, J. Hydr. Eng. 109 1740–1754.
8. Stamou, A.I., Adams, E.A., Rodi, W., 1989. Numerical modelling of flow and settling in primary rectangular clarifiers. J. Hydraul. Res. IAHR 27 (5), 665–682.
9. Adams, E.W., Rodi, W., (1990). Modelling flow and mixing in sedimentation tanks. J. Hydraul. Eng. 116 (7), 895–913.
10. Lyn, D.A., Stamou, A., Rodi, W., (1992). Density currents and shear induced flocculation

ISSN: 2277-9655
Scientific Journal Impact Factor: 3.449
(ISRA), Impact Factor: 2.114

in sedimentation tanks. J. Hydr. Eng. ASCE 118 (6), 849–867.

11. Frey, P.H., (1993). Hydrodynamics fields and solid particle transport in a settling tank. J. Hydr. Res. 31 (6), 763–776.
12. Van derWalt, J.J., (1998). Is a sedimentation tank really that simple? In: Proceedings of the WISA Biennale Conference and Exhibition, Cape Town, South Africa.
13. Wilcox D.C.,(1998). Turbulence Modeling for CFD, DCW Industries, Inc., La Canada, California,.
14. Chu, C.P., Lee, D.J., Tay, J.H., (2005). Floc model and intraflocflow.Chem. Eng. Sci. 60, 565–575.
15. Deininger, A., Holthausen, E., Wilderer, P.A., (1998). Velocity and solids distribution in circular secondary clarifiers: full scale measurements and numerical modelling. Water Res. 32 (10), 2951–2958.