*Journal of Water Sciences Research,*
*ISSN: 2251-7405 eISSN: 2251-7413*
*Vol.5, No.1, Spring 2013, 55-63, JWSR*

JWSR

# *Investigating the Effect of Inlet and Outlet Location on Flow Field and Sedimentation Pattern of a Rectangular Settling Basin Using CCHE2D*

Mohammad Reza Majdzadeh Tabatabaie[1], Mohammad Reza Pirestani [2], Saeideh Alimohamadi[3*]

1. Assistant Professor, Department of Civil Engineering, Power and Water University of Technology, Tehran, Iran.
2. Assistant Professor, Department of Civil Engineering, Islamic Azad University-South Tehran Branch, Tehran, Iran.
3*. Graduate M. Sc., Islamic Azad University-South Tehran Branch, Tehran, Iran

Received: 30 May 2013 Accepted: 19 June 2013

### *ABSTRACT*

Water obtained from any source may contain impurities induced by erosion, dissolved minerals, dissolved gases and material produced from organic matter decomposing. If the amount of these substances exceed a certain limit, not only the water would not be potable, but the water containing the material will not be suitable to be used in most applications, or at least before treatment cannot be applied in everyday use. Sediment particles separation from flow via gravity in settling tanks and basins is one of the effective solutions in water treatment, urban water delivery projects and prevention of sediment entrance into irrigation networks. Construction cost of a settling basin is about one third of the total expenditure involved in constructing a water treatment plant. So a great economy may be achieved by increasing settling efficiency. Herein this paper by applying CCHE2D numerical model, the effect of geometric parameters such as inlet and outlet locations on flow field and sedimentation pattern in a rectangular settling basin is investigated. Results show that some variations in flow field and eddies formation and hence, sedimentation pattern takes place in the basin. Results also confirm that the model could appropriately simulate flow field and sedimentation pattern with a reasonable error percentage.

### *Keywords*

settling basin; flow field; CCHE2D numerical model; sedimentation pattern; inlet and outlet location.

## 1. Introduction

Sedimentation is the process which has widespread use in separation of inorganic materials from water. In fact sedimentation is one of the main processes used in water industry, for industrial and domestic wastewater treatment (Firouzabadi et al., 2007). Construction cost of a settling basin is about one third of the total expenses of a water treatment plant construction which encompasses a significant percentage of costs (Firouz abadi and Tamayol, 2004). Thus by increasing deposition efficiency one can get a significant saving in various stages. By correct design and increasing efficiency of settling basins not only significant savings will be achieved in construction and operation costs, but also lifetime of other components of water treatment plant will increase.

First investigations on basins returns back to Dobbins (1944) and Camp (1946), and despite the awareness about rotating and

* Corresponding Author Email: (saeideh.am@gmail.com)

turbulent zones in flow, suitable modeling is not done in this case. Selik et al. (1985) used the two equation standard linear k-ε model for modeling. Stamou and his group in 2001 while using standard k-ε model and flow through curves optimized a basin in Athens.

Tamayol and Firoozabadi (2006) by using FTC method or RTD investigated the efficacy of settling basins. They used a two dimensional simulation by applying the Euler-Lagrange equations to study the effect of baffles of the various forms and in different locations on efficiency of the settling basin and determined the appropriate location of the entrance and the baffle in the basin.

Goula et al. (2008) used CFD to investigate the effect of inlet baffle on the efficiency of particle sedimentation in settling basins [6].

Stamou (2008) while using the standard k - ε and FTC curves (Flow Through Curves) optimized the main basins of water supply network in Athens.

Rostami et al. (2011) investigated the effects of inlet located at various depths on the flow pattern in the primary settling basins by using numerical simulations. They used Flow-3D Computational Fluid Dynamics numerical model.

Golriz et al. (2011) numerically investigated the effect of baffles on the performance of municipal settling basins using a 3D numerical model. They simulated 3 different inlet and outlet baffles.

The scope of this paper is to investigate the effect of three different alternatives for inlet and outlet locations on flow pattern and sedimentation in a rectangular settling basin.

## 2. CCHE2D Numerical Model

Numerical simulation is done by using CCHE2D (two dimensional flow and sediment transport model) model (Jia &Wang 1999, Wu 2001). This model has been presented by NCCHE (National Center for Computational Hydro Science and Engineering, USA), is a 2-D depth averaged hydrodynamic and sediment transport model. The model is used to predict flow patterns in river and associated scour in beds or banks for uniform and non-uniform sediment transport. Both the $k-\varepsilon$ depth averaged turbulence model and the eddy viscosity are available. The general process which is used in this model is shown schematically in Figure 1.

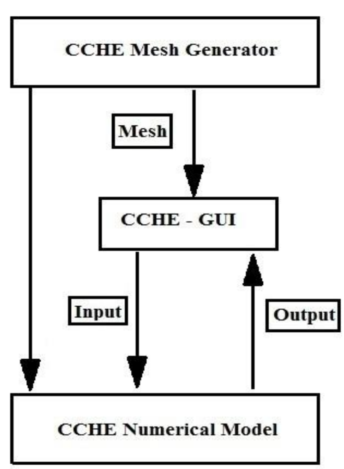


Fig. 1. schematic view of the CCHE2D software process

**a. Governing Equations**

The depth integrated two-dimensional equations are solved in CCHE2D model:

Continuity Equation:

$$\frac{\partial Z}{\partial t}+\frac{\partial (hu)}{\partial x}+\frac{\partial (hv)}{\partial y}=0 \qquad (1)$$

Momentum Equations:

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=$$
$$-g\frac{\partial z}{\partial x}+\frac{1}{h}\left[\frac{\partial (h\tau_{xx})}{\partial x}+\frac{\partial (h\tau_{xy})}{\partial y}\right]-\frac{\tau_{bx}}{\rho h}+f_{cor}\,v \qquad (2)$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=$$
$$-g\frac{\partial z}{\partial y}+\frac{1}{h}\left[\frac{\partial (h\tau_{yx})}{\partial x}+\frac{\partial (h\tau_{yy})}{\partial y}\right]-\frac{\tau_{by}}{\rho h}-f_{cor}\,u \qquad (3)$$

Where u and v are the depth-integrated velocity components in the x and y directions, respectively, g is the gravitational Acceleration, Z is the water surface elevation, ρ is water density, h is the Local water depth, $f_{cor}$ is the Coriolis parameter; $\tau_{xx}$, $\tau_{xy}$, $\tau_{yx}$ and $\tau_{yy}$ are the depth integrated Reynolds stresses; and $\tau_{bx}$ and $\tau_{by}$ are shear stresses on the bed surface.

CCHE2D is a free code and precise details about the governing equations and their characteristics can be found in Wu (2001).

**3. Material and Methods**

In order to calibrate the model, experimental results of Sameh Kantoush et al. (2008) experiments in Hydraulic Structures Laboratory in Swiss Federal Institute of Technology (EPFL) was used. The setup consists of a rectangular inlet channel, 0.25 m wide and 1.0 m long, made of PVC, a rectangular shallow basin with inner dimensions of 6.0 m long and 4.0 m wide, a rectangular outlet channel 0.25 m wide and 1.0 m long, and finally a flap gate 0.25 m wide and 0.30 m high at the end of the outlet. A schematic view of the basin used by Kantoush et al. (2008) is shown in Figure 2.

In order to measure the velocity in the laboratory, devices namely Ultrasound Velocity Profiler (UVP) and Large Scale Particle Image Velocimetry (LSPIV) were used. A schematic view of the flow pattern measured by these two experimental tools is shown in Figure 3.

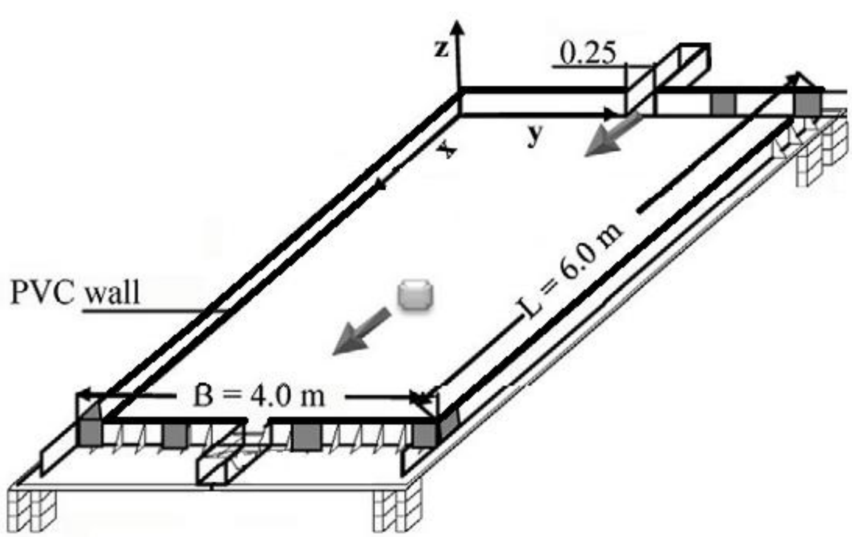


Fig. 2. A schematic view of the basin used by Kantoush et al. (2008)

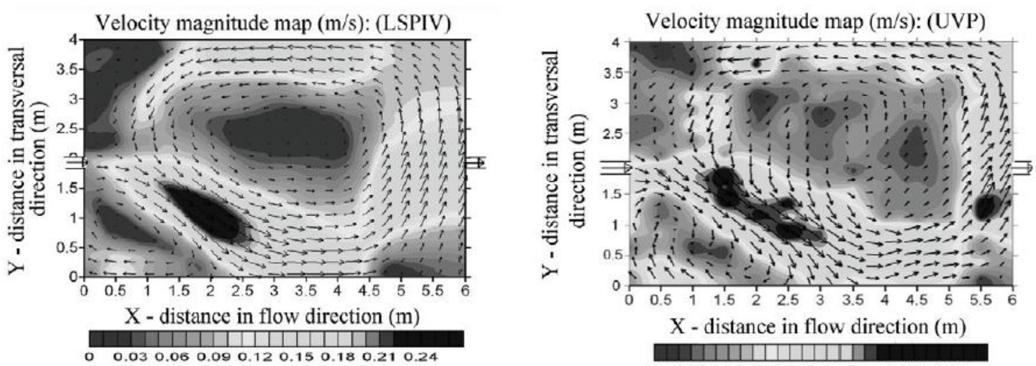


Fig. 3. Flow field pattern measured by LSPIV and UVP (Kantoush et al., 2008)

## 4. Initial and Boundary Conditions

Initial and boundary conditions for the flow include initial bed level, initial water surface and bed roughness that are entered as 0, 0.2 and 0.015 in SI units, respectively. Bed roughness is a parameter that controls the simulation process and is changed for the calibration until the software output data would be in good agreement with the measured results. Input discharge and sediment concentration were considered 0.007 $m^3/s$ and of 3 $kg/m^3$, respectively. A parabolic eddy viscosity model is used as the turbulence closure model.

## 5. Numerical Simulation

In order to investigate the effect of inlet and outlet locations on the flow field and hence sedimentation in the basin, 3 different alternatives are examined namely: 1- inlet and outlet are located in the middle of the inlet and outlet sides of the basin, respectively, 2- inlet is located on the left corner of the inlet side while the outlet is located on the right corner of the outlet side of the basin and finally 3-inlet and outlet are located on the upper and lower corners of the inlet side of the basin, respectively.

### a. Model Calibration for centre line velocity magnitude

The first basin alternative which was used by Kantoush et al. is employed for model calibration. Considering the required time duration for flow stabilization inside the basin, results of Kantoush et al. (2008) experiments as 4.5 hours were used for simulation.

Computational error percentage for velocity magnitude in the centre line of the basin was 25 percent in comparison to experimental measurements by using LSPIV device. In the points near the inlet and outlet of the basin, because of the sudden change from 1 m to 4 m of the width, it seems that the software is not accurate enough. So by omitting first and last 1 meter of the basin, the average error percenttage would reach 15 percent which shows a good agreement with experimental results. Figure 4 shows the computed axial velocity magnitude at the basin centerline.

### b. Sensitivity Analysis

After achieving an acceptable model calibration, sensitivity analysis is done on inlet and outlet locations as described in section 5. Investigated locations and simulation results for velocity magnitude are shown in Figure 5 to Figure 7.

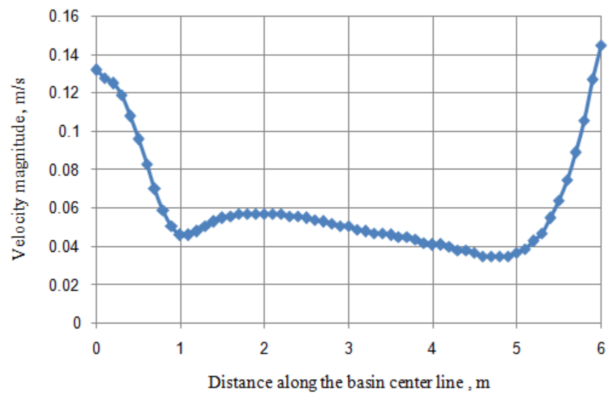


Fig. 4. Computed velocity magnitude at the basin centerline using CCHE2D for case1

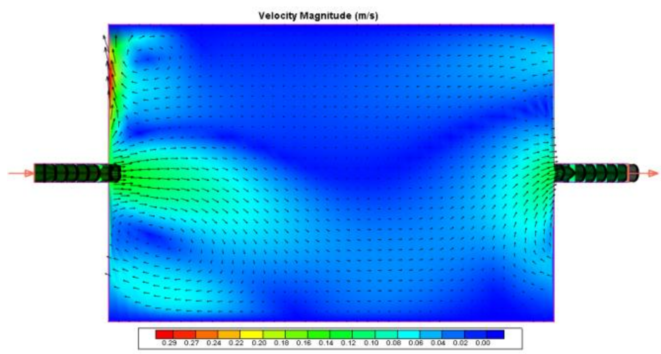


Fig. 5. Flow pattern for case 1

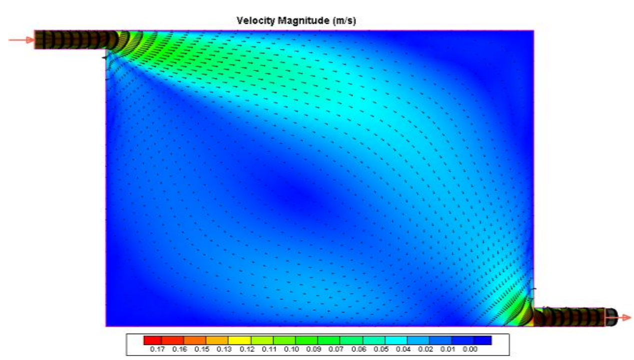


Fig. 6. Flow pattern for case 2

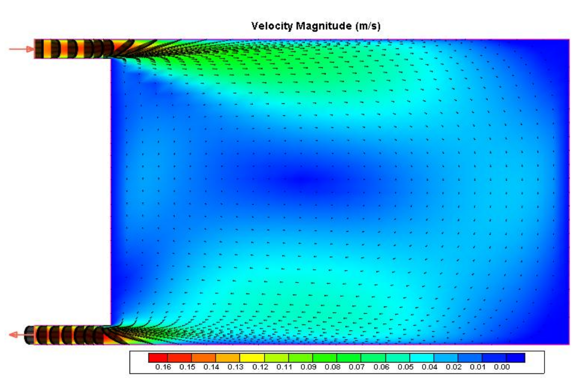


Fig. 7. Flow pattern for case 3

## 6. Results and Discussion

### 6.1. Case 1

In some areas within the basin, longitudinal velocities have negative values which are in opposite direction to the main flow toward the outlet and will cause the sediments reversal to the basin and vortex formation. Vortices will cause short circuiting, increase the dead areas and change the mixing ratio that prevents sedimentation and reduces the basin performance.

Transversal velocities in the upstream upper corner of the basin take high positive values that in combination to the longitudinal velocities will cause the vortex formation in the upper corner of the basin.

Direction and magnitude of the velocity is achieved by combining the longitudinal and transversal velocities in the basin. It can be seen in case 1 that after jet issuance, the main flow tends to go towards the right hand side generating a large and stable main vortex, rotating anticlockwise, and two small vortexes rotating clockwise in the two upstream corners of the basin.

### 6.2. Case 2

By investigating longitudinal velocity contours it can be seen that by changing the inlet and outlet location in this case, velocity vectors become more uniform and flow after entering the basin moves directly toward the outlet. In small areas negative velocities are observed that help in the formation of a central vortex.

Transversal velocities in this case have very small changes from the degree of 0.01 which has a mild effect on the flow overall pattern.

In case 2, a central vortex is formed in the middle of the basin. By changing the location of inlet and outlet, the small vortexes in the previous state are removed from the flow pattern and flow turbulence is reduced. Sedimentation pattern seems to be more concentrated in this case.

### 6.3. Case 3

By investigating longitudinal velocity contours it can be seen that by changing the inlet and outlet location in this case, velocity vectors become more uniform and flow after entering the basin passes a circular direction and leaves the basin. In the lower half of the basin negative longitudinal velocity vectors were observed which is because of flow deviation toward the outlet. These negative velocities help in the formation of a central vortex.

Transversal velocities in this case have very small changes from the degree of 0.01 which has a mild effect on the flow overall pattern.

By combining the longitudinal and transversal velocities in the basin it can be seen that a central small vortex is formed in the middle of the basin. By changing the location of the inlet and outlet in case 3, small vortexes formed in case 1 are removed from the basin and thus, flow turbulence is reduced.

Shear stress is always one of the effective parameters in sedimentation and scour phenomena. Researches show that zones having the maximum amounts of shear stress are the most vulnerable zones for bed erosion, and zones having less shear stress are the most suitable places for sedimentation. So, probable zones for sedimentation can be predicted by investigating shear stress distribution. CCHE2D calculates shear stress using equations 4 to 6.

$$\tau_{bx} = \frac{1}{h^{1/3}} \rho g n^2 u U \tag{4}$$

$$\tau_{by} = \frac{1}{h^{1/3}} \rho g n^2 v U \tag{5}$$

$$U = \sqrt{u^2 + v^2} \tag{6}$$

In these equations ρ is water density, n is manning roughness coefficient and h is local water depth.

As can be seen from Figure 8, there is a small area of high shear stress in the upper left corner of the basin in comparison to the other areas. Increased shear stress in this region is due to high velocity gradient that can lead to sediment wash out from the bed surface. In cases 2 and 3, as shown in Figures 9 and 10, respectively, because the high velocity gradient in the upper left corner of the basin is removed, the area with high shear stress is omitted and it seems that the probability of sediments being washed from the bed is reduced and as a result, performance of the settling basin is increased.

Shear stress simulation results in the basin for different inlet and outlet locations are presented in Figures 8 to 10.

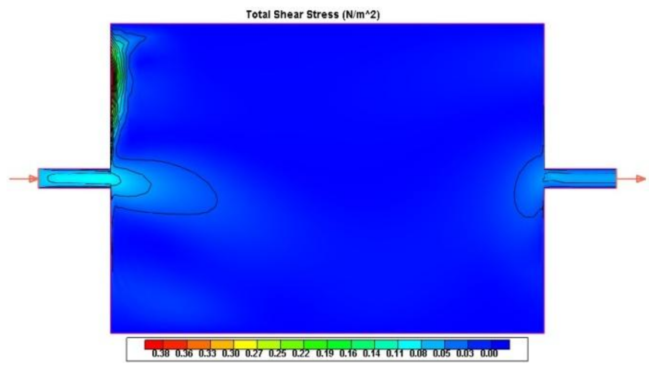


Fig. 8. Shear stress for case 1

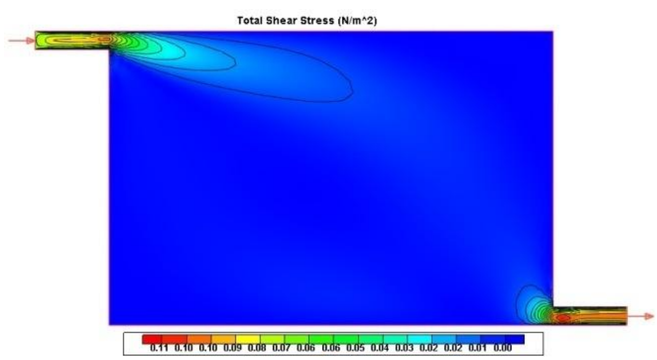


Fig. 9. Shear stress for case 2

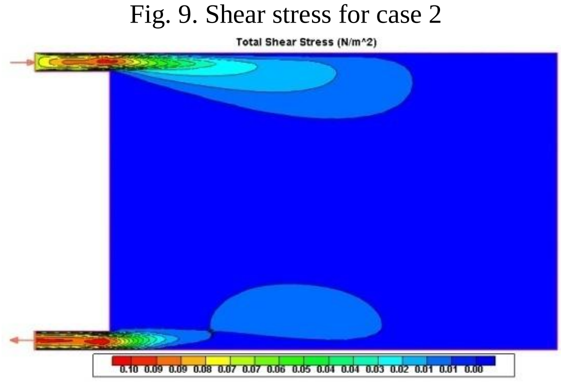


Fig. 10. Shear stress for case 3

## 7. Conclusions

In this paper the effect of inlet and outlet locations on flow pattern in a rectangular settling basin is investigated by using numerical modeling of CCHE2D and the following results are obtained:

1- By changing inlet and outlet locations, changes were made in vortexes inside the basin so that in cases 2 and 3 just a main central vortex was formed, while in case 1 smaller eddies were formed at two upstream corners of the basin. This shows the importance of inlet and outlet location on flow and sedimentation patterns.

2- Flow pattern in case 2 comparing to other cases spreads on a smaller surface of the basin and sedimentation seems to have a more concentrated condition comparing to other cases.

3- Calculation of bed shear stress shows that changes in inlet and outlet locations have significant effect on sedimentation patterns.

4- Results also confirm that CCHE2D numerical model can appropriately simulate 2-D flow field and sedimentation pattern with a reasonable error percentage.

5- 2-D simulation in comparison to the 3-D one is significantly time saving and gives reasonable results.

6- This software cannot model flow field near inlet and outlet boundaries especially when the geometric condition changes greatly. This can be attributed to the numerical solution or the calculations related to the energy loss. This is one of the dis-advantages of the model.

7- In order to have a more precise invest-tigation on shear stress distribution and to predict sedimentation pattern, doing simulations by using different entrance discharges, and also sensitivity survey for turbulence closure theories are suggested.

## References

Camp, T. R. (1946), “Sedimentation and the Design of Settling Tank,” Transaction, ASCE, Vol. 3, pp. 895-936.

CCHE and CCHE-GUI (2006), Graphical Users Interface For NSSH Model, User`s Manual, Version 3.

Dobbins, W.E. (1944), Effect of Turbulence on sedimentation. Transaction, ASCE, Vol. 109, No. 2218, pp. 129 -656.

Firouz Abadi, B. (2007), Investigating baffle effect on flow pattern in primary rectangular sedimentation basins. 15th annual mechanics engineering conference, Amir Kabir industrial University, Tehran, Iran (in Persian).

Golriz, F., Salehi Neyshaboori, A., (2011), Investigation on the effect of the baffle on the performance of the municipal sedimentation basins, Tenth Iranian Hydraulic Conference.

Goula et al. (2008). A CFD Methodology for the Design of Sedimentation Tanks in Potable Water Treatment, Chemical Engineering Journal 140, pp. 110-121.

Rostami. F. et al. (2011), Numerical Modeling of Inlet Aperture Effects on Flow Pattern in Primary Settling Tanks. Applied Mathe-matical modeling, Vol. 35, pp. 3012-3020.

Sameh A.Kantoush, Erik Bollaert, Anton J. Schleiss (2008). Experimental and Numerical Modeling Of Sedimentation in a Rectangular Shallow Basin. International Journal of Sediment Research, Vol. 23, pp. 212-232.

Selik, I. Rodi, W. and Stamou, A. I. (1985), Prediction of Hydro-Dynamic Charac-teristics of Rectangular Settling Tanks,” Int. Symposium of Refined Flow Modeling and Turbulence Measurements, IOWA, USA.

Stamou, A. I. (2008). Improving the Hydraulic Efficiency of Water Process Tanks Using

CFD Models. Chemical Engineering and Processing, 47, pp. 1179-1189.

Tamayol, A. and FiroozAbadi, B. (2006). Effects of Turbulent models and Baffle Position on Hydrodynamics of Settling Tanks. Scientia of Iranica Journal 13(3), pp. 255-260.