Numerical Simulation of Influence of Inlet Configuration on Flow Pattern in Primary Rectangular Sedimentation Tanks

Mahdi Shahrokhi, Fatemeh Rostami, Md Azlin Md Said and Syafalni

School of Civil Engineering, Universiti Sains Malaysia, 14300 Nibong Tebal,
Seberang Perai Selatan, P. Penang, Malaysia

Abstract: Design of primary settling tank has historically been done more empirically than rationally. The main reason for this is a lack of understanding of what pollutants primary settling tank are capable of removing. Inlets should be designed to dissipate the kinetic energy or velocity head of the mixed liquor. They should be able to distribute the flow equally in vertical and horizontal directions. Inlets should also be designed to prevent short-circuiting. Flow in primary settling tank is simulated by means of computational fluid dynamic (CFD). The fluid is assumed incompressible and non-buoyant. Two-dimensional simulation for one-phase flow has been done. The RNG turbulent model was solved with the Navier-Stockes equations. The flow pattern properties in the settling tank include the velocity profiles, the flow separation area and kinetic energy. In this study, the effect of inlet configurations on these characteristics is considered. Due to this object, rectangular, slope and curved shape of inlet slot are suggested.

Key words: Sedimentation Tanks · Inlet Structure · Kinematic Energy · Numerical Simulations · VOF

INTRODUCTION

Removing suspended particles from water by gravity is known as sedimentation. This method is an integral part of any water and wastewater treatment plant and they have 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. In these tanks, the low speed turbid water will flow through the length of the tank and suspended particle have enough time to settle. Finding new and useful methods to increase hydraulic efficiency is the objective of many theoretical, experimental and numerical studies. Sedimentation tanks can be rectangular with horizontal flow or circular where an upflow pattern results. In rectangular tanks, influent enters the basin at the inlet. Energy dissipation is the main objective in designing a primary clarifier inlet. Energy of influent must be dissipating at the inlet zone by selecting the best position and configuration of inlet or using the baffles in the inlet zone [1].

Enlarging the size of the inlet zone and using the inlet energy for flocculation can improve suspended solids

removals. Impinging flow streams against one another is an effective way of promoting flocculation [2]. Smaller inlet apertures made larger eddies and, thus, removal efficiency decrease [3]. Density effects and potential energy maybe decreased by a low position of inlet, as proposed by some researches [1, 4-6]. The best position for the inlet is to be placed somewhere in the mid depth of the tank and the inlet from the bottom is better than the inlet from the surface [5]. Flocculation and avoiding floc break up was the main objective of Larsen [7] inlet design developed in pure-water model tests by evaluating velocity profiles in place of FTCs.

Goula *et al.* [8] studied on the effect of inlet baffle height on flow pattern and particles trajectories throughout the tank or at the exit. The authors found that the baffle affect on the inlet section and near the bottom of the tank. It seems that the extended baffle provide better influent mixing and isolation between the tank influent and effluent than the short baffle in the original tank design, thus significantly enhancing sedimentation. The extended baffle increases the kinetic energy and the dissipation rate in the inlet baffle in region and, consequently, weakens the currents in this region. Having

Corresponding Author: Md Azlin Md Said, School of Civil Engineering, USM, Engineering Campus,
14300 Nibong Tebal, Seberang Perai Selatan, P. Pinang, Malaysia.
Tel: 04-5995834, 6202, Fax: 04-5941009.

two or three slots serving as inlets is better compared to having only one aperture because uniform flow was generated in shorter distances and the turbulence kinetic energy and volume of circulation zone was considerably low in these cases [9].

Governing Equation: The governing equations are general mass continuity and momentum. The turbulence model is likewise solved with these equations to calculate the Reynolds stresses. The mass continuity equation for fluid is a simple form. As the flow pattern is assumed to be two-dimensional, two momentum equations in the x and z directions are solved. The general mass continuity equation is as follows:

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

where (u,w) are the velocity components in the length and height (x,z) directions. The momentum equation for the fluid velocity components in the two directions are the Navier-Stokes equations, which are as follows:

$$\frac{\partial u}{\partial t} + \frac{1}{V_F}\left\{ uA_x \frac{\partial u}{\partial x} + wA_z \frac{\partial w}{\partial z} \right\} = -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + f_x$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_F}\left\{ uA_x \frac{\partial w}{\partial x} + wA_z \frac{\partial w}{\partial z} \right\} = -\frac{1}{\rho}\frac{\partial P}{\partial z} + G_z + f_z \qquad (2)$$

where V_F is the fractional volume of the flow in the calculation cell, ρ is the fluid density, P is pressure and G_x and G_z are body accelerations. Meanwhile, f_x and f_z are viscous accelerations, which for a variable dynamic viscosity μ are as follows:

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

$$\rho V_F f_x = wsz - \left\{ \frac{\partial}{\partial x}(A_x \tau_{xz}) + \frac{\partial}{\partial z}(A_z \tau_{zz}) \right\} \qquad (3)$$

where:

$$\tau_{xx} = -\frac{2}{3}\mu\left\{2\frac{\partial u}{\partial x} - \frac{\partial w}{\partial z}\right\}; \tau_{zz} = -\frac{2}{3}\mu\left\{2\frac{\partial w}{\partial z} - \frac{\partial u}{\partial x}\right\}; \tau_{xz} = -\mu\left\{\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right\}$$

$$(4)$$

The free surface profile is calculated using the VOF function, F(x, z, t), with the VOF method. The computational cells can be classified as empty, full, or partially filled with fluid. Hence, the fluid fraction for each cell in the computational domain varies from zero to one, depending on the quantity of fluid in the cells.

Thus, when F = 1, show a cell filled by fluid and F = 0 correspond void region. This function presents the volume of fluid per unit volume and satisfies the following equation

$$\frac{\partial F}{\partial t} + \frac{1}{V_f}\left\{ \frac{\partial}{\partial x}(FA_x u) + \frac{\partial}{\partial z}(FA_z w) \right\} = 0 \qquad (5)$$

In one-phase flow, F shows the volume fraction filled by the fluid. Voids are regions without fluid mass that have uniform pressure appointed to them. Physically, they represent regions filled with vapor or gas whose density is negligible compared to fluid density

To consider the turbulence effects on the flow field, the renormalization group (RNG) model was utilized. The RNG model is a powerful turbulent model applied particularly for flows with curved streamlines. It concerns statistical methods to derive the averaged equations for turbulence quantities, such as turbulent kinetic energy and its dissipation rate. The RNG-based methods rely little on empirical constants while setting a framework for the derivation of a range of models at different scales [10, 11]. The RNG model uses equations similar to those for the k-∈ model. However, the equation constants found empirically in the standard k-∈ model are derived explicitly from the RNG model. The turbulence kinetic energy and its rate of dissipation (i.e., k and ∈, respectively) are obtained from the following transport equations:

$$\frac{\partial k_T}{\partial t} + \frac{1}{V_F}\left\{ uA_x \frac{\partial k_T}{\partial x} + wA_z \frac{\partial k_T}{\partial z} \right\} = P_T + Diff_T - \varepsilon_T \qquad (6)$$

where k_T is the turbulent kinetic energy and P_T is the turbulent kinetic energy production. The diffusion term is as follows:

$$P_T = \frac{\mu}{\rho V_F}\left\{ 2A_x\left(\frac{\partial u}{\partial x}\right)^2 + 2A_z\left(\frac{\partial w}{\partial z}\right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x}\right)\left(A_z \frac{\partial u}{\partial z} + A_x \frac{\partial w}{\partial x}\right) \right\} \qquad (7)$$

The diffusion term is:

$$Diff_T = \frac{1}{V_F}\left\{ \frac{\partial}{\partial x}\left(\upsilon_k A_x \frac{\partial k_T}{\partial x}\right) + \frac{\partial}{\partial z}\left(\upsilon_k A_z \frac{\partial k_T}{\partial z}\right) \right\} \qquad (8)$$

where υ_k is the diffusion coefficient computed based on the local value of the turbulent viscosity. An additional transport equation must be solved for the turbulent dissipation, ε_T:

$$\frac{\partial k_T}{\partial t} + \frac{1}{V_F}\left\{ uA_x \frac{\partial \varepsilon_T}{\partial x} + wA_z \frac{\partial \varepsilon_T}{\partial z} \right\} = \frac{C_1 \varepsilon_T}{k_T}(P_T + C_3.G) + Diff_\varepsilon - C2.\frac{\varepsilon_T^2}{k_T} \qquad (9)$$

The diffusion of dissipation, *Diff*$_\varepsilon$, is as follows:

$$Diff_\varepsilon = \frac{1}{V_F}\left\{\frac{\partial}{\partial x}\left(v_\varepsilon A_x \frac{\partial \varepsilon_T}{\partial x}\right) + \frac{\partial}{\partial z}\left(v_\varepsilon A_z \frac{\partial \varepsilon_T}{\partial z}\right)\right\} \qquad (10)$$

Here C_1, C_2 and C_3 are all dimensionless parameters that are equal to 1.42, 1.68 and 0.2, respectively, for the RNG model [10, 12].

In this study, a Flow-3D flow solver module that used finite difference schemes for structured meshes was used to simulate the settling tank. The flow field is divided into fixed rectangular cells.

Numerical Model: In this study, FAVOR (Fractional Area/Volume Obstacle Representation) method was used to investigate the geometry in finite volume mesh. This technique developed by Hirt and Sicilian [13]. The FAVOR determines the obstacles in a calculation cells with a fractional value between zero to one as obstacle fills in the cell. Geometry of obstacle are inserted in the mesh by setting the area fractions on the cell faces along with the volume fraction open to flow [14]. This way makes independent geometry formation on grid and then the complex obstacle can be generated.

In present study, the available Computational Fluid Dynamics (CFD) program Flow-3D, developed by Flow Sciences, was used for the numerical simulation. This CFD code uses volume (or finite difference) scheme for structured meshes to solve the RANS equations. The flow field is divided into fixed rectangular cells. The average values of flow parameters such as pressure and velocity for each cell are computed at discrete times. The new velocity in each cell is calculated from the coupled momentum and continuity equation using previous time step values in each centre of face of cells. The mass and momentum equations joined by means of velocity can be used to obtain an equation for the pressure term. The pressure term is computed and modified using the estimated velocity to satisfy the mass equation. With the computed velocity and pressure for later time, remaining variables are estimated including turbulent transport, density advection and diffusion and wall function evaluation. When the flow filed is turbulent, the computation becomes more complex. Due to this, the Reynolds-Averaged Navier-Stokes (RANS) equations are prevalently used. It is a modified form of the Navier-Stokes equation inclusive the Reynolds stress term, which approximates the random turbulent fluctuations by statistics.

Verification Test: To verify the computational simulation of settling tanks, the experimental conditions used by Imam *et al*. [3] were considered. The experiment was conducted in a settling tank with length of 73 cm, depth of 11.9 cm, an opening inlet of 6.9 cm and a flow rate of 109 cm3/s/cm. Sediment particles were imported in the sedimentation tank to measure the efficiency of the tank [3]. In this study, the numerical model was applied to simulate this basin using a uniform rectangular mesh. The cell size in each direction is about 8 mm. The RNG turbulence model was chosen to calculate turbulence effects on the flow field. In this simulation, the flow is clear and has no particles.

Fig. 1 shows a comparison between the results of the numerical models and experiments by Imam *et al*. [3]. Clearly seen that form Fig. 1, which numerically results show a good prediction to the experimental result.
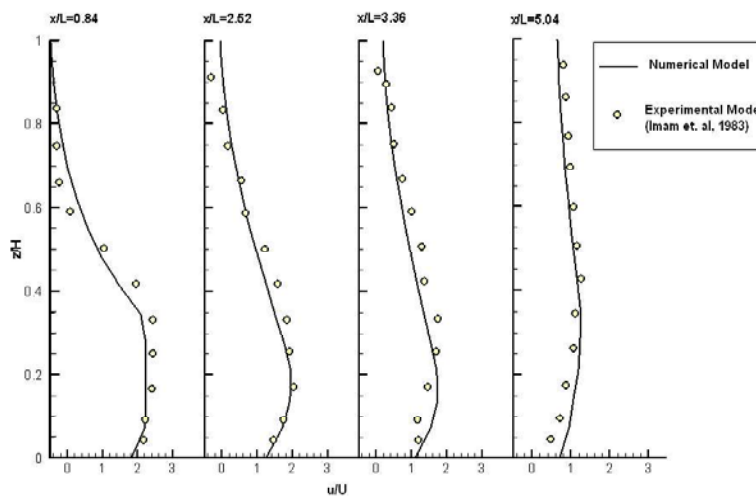

Fig. 1: Comparison of velocity distribution from numerical simulation with experimental result of Imam *et al*. [16]
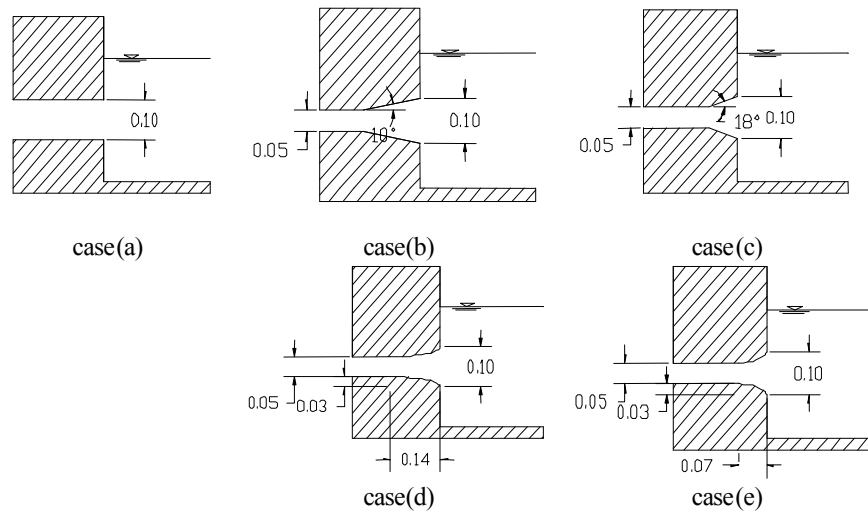
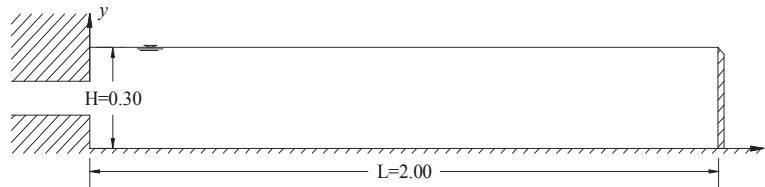Fig. 2: Five configurations of inlet apertures

Fig. 3: Schematic shape of sedimentation tank

Due to low concentration in the upper zone of the flow, the simulation results converge with those from the experimental test of Imam *et al*. [3]; however, the particles affect the flow field in the lower section of the flow duo to sedimentation. The results of clear water simulation differ from the experimental data, indicating that the computational model can accurately simulate flow patterns in the basin.

Modeling Approach: A sedimentation tank, with a volume of 2.0m length, 0.5m width and 0.3m depth was modelled in present study. In this paper, the effect of inlet shape on the flow pattern in the settling basin was investigated. Some researchers suggested that the best location of inlet aperture in primary sedimentation tank is somewhere in the middle of depth [1, 5, 6]. Base on this idea, five configurations (a to e) have been considered in this study (Fig. 2).

A schematic view of sedimentation tanks was showed in Fig. 3. To reduce computational power requirements, the rectangular sedimentation tank was modelled in 2-dimensional. The major assumption in the development of the model is that the flow pattern is the same for all cross sections of basin; therefore, a 2D model can be applied to properly simulate the general

characteristics of the hydrodynamic processes in the tank. As a first step, a mesh was generated across the sedimentation tank. A grid dependency study was executed to eliminate errors due to the roughness of the grid and to determine the best agreement between simulation accuracy, numerical stability, convergence and computational time and the mesh was finest where velocity gradients are expected to be larger. The selected grid was composed of 18560 rectangular elements.

As stated before RNG turbulence model was used to account for turbulence, since this model is meant to describe better for flow with curvature streamlines. The converged solution was defined as the solution for which the normalized residual for all variables was less than 10^{-5}. The water flow rate is 1.00 L/s and the depth of water in the tank is about 0.30m. The aperture height is 0.1m for all cases.

Boundary Conditions: To simulate a given flow, it is important that the boundary conditions accurately represent what is physically occurring. The inlet condition was specified as a plug flow of water at 1.00 L/s and the outlet was indicated as outflow boundary condition and the atmospheric pressure at the top. No slip wall function is assumed for all rigid boundary and uniform velocity

profile in inlet for all cases. Water surface profile is calculated by VOF method. In addition, the symmetry condition is applied for zero gradient perpendicular to the boundary. All obstacles and walls are assumed no slip. No slip is defined as zero tangential and normal velocities (u = w = 0). With no-slip boundary, it is assumed that a law-of-the-wall type profile exists in the boundary region. A 1/7 power law is used to approximate the logarithmic law-of-the-wall expression, which modifies the wall shear stress magnitude [15].

RESULTS AND DISCUSSION

Flow Pattern: The performance of sedimentation tanks depends on the flow pattern and the mixing regime in the tank. The inlet structure in sedimentation tanks must provide a uniform flow pattern to minimizing the circulation zones. Consequently, the determination of flow and mixing characteristics is indispensable for the prediction of the tank efficiency. Fig. 4 shows the calculated streamlines for the different configurations of inlet opening. Usually, the flow pattern is characterized by a large recirculation region straddling a large part of the tank from top to bottom. Case (a) has the largest circulation zone and changing the configuration of inlet opening, improves these situations and the circulation zone become smaller with this modify. There are no significant differences between modified cases and all of them could diminish the volume of death zones.

Velocity Profiles and Kinetic Energy: The velocity in the sedimentation tanks must be as low as possible and velocity profile must be uniform to get high rate of sediment settled. The inlet structure should be design to reduce the velocity and produce more uniform velocity profiles in the tanks. Fig. 4 shows the x-velocity vectors; Fig. 5 shows x-velocity contour and Fig. 6 displays z-velocity for all cases.
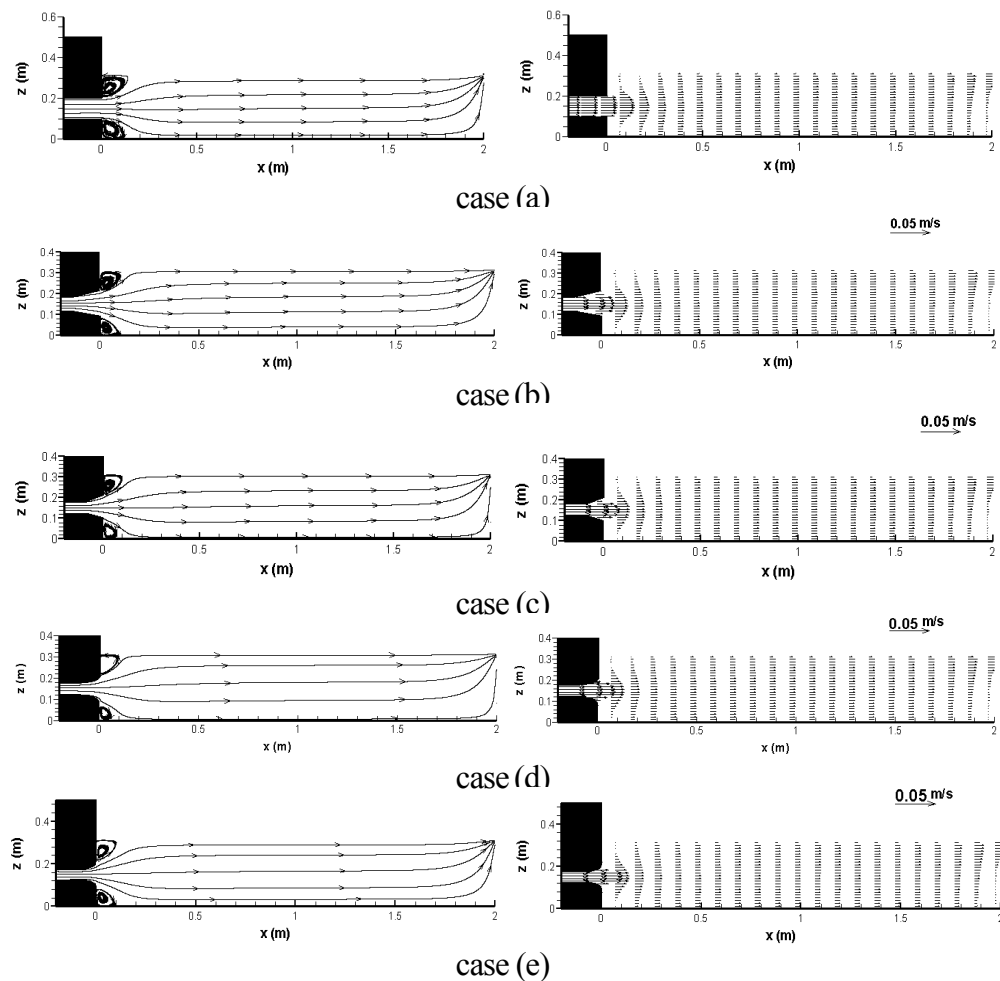

case (a)

case (b)

case (c)

case (d)

case (e)

Fig. 4: Streamlines at x-velocity vectors for different inlet conditions

case (a)

case (b)

case (c)

case (d)

case (e)

Fig. 5: *x*-velocity contour in the tank

case (a)

case (b)

case (c)

case (d)

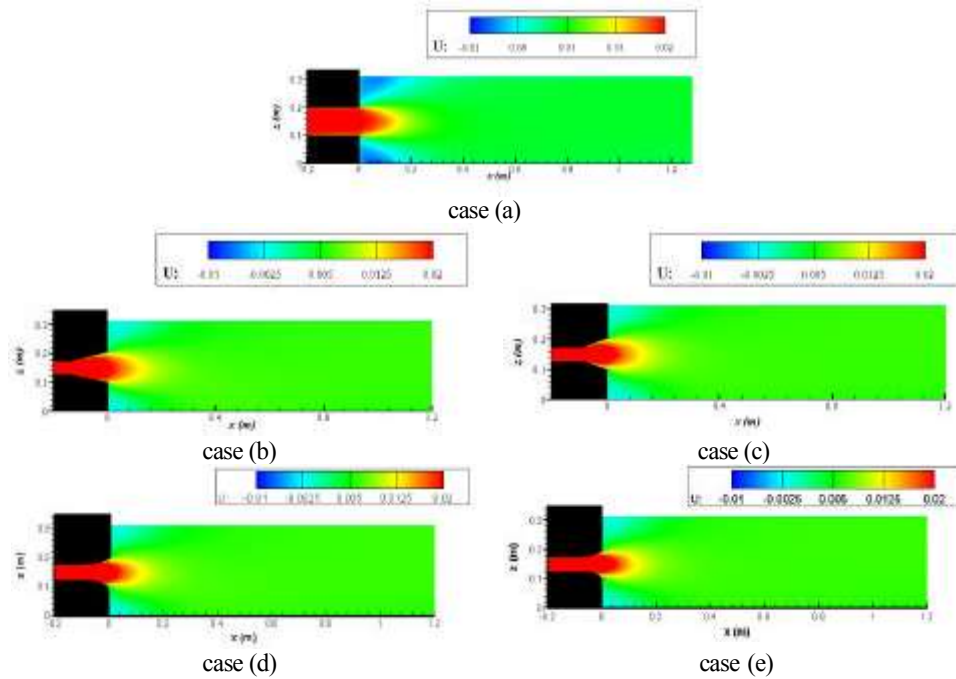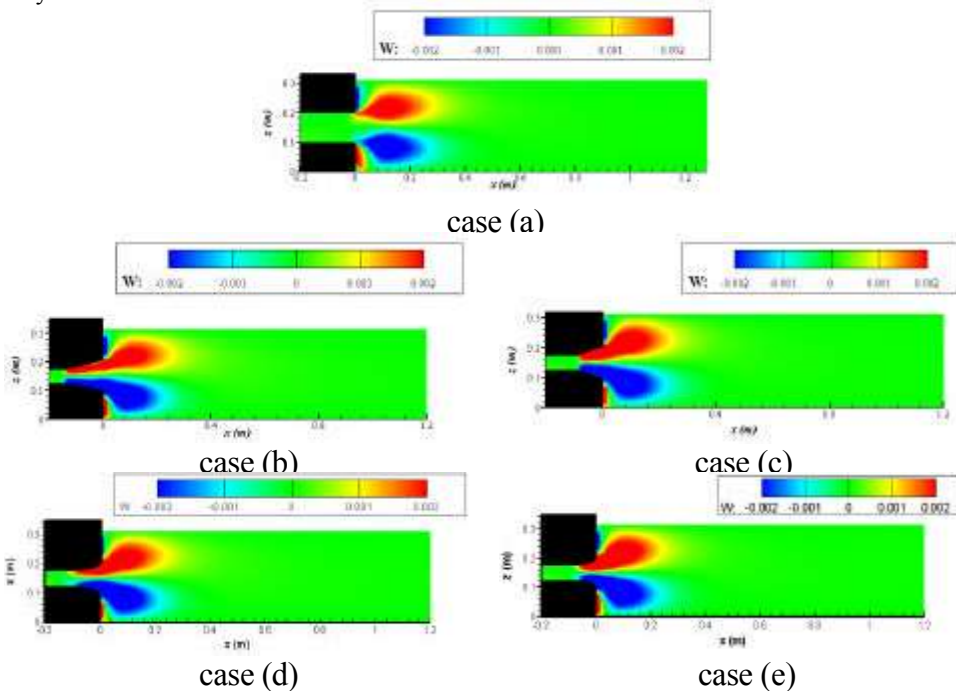case (e)

Fig. 6: *z*-velocity contour in the tank

Fig. 5 shows the x-velocity contour for the studying cases. There are two strong circulation volumes with high negative velocity in case (a). The circulation zones become smaller and weaker at the customized cases. The expanded entrance reasons to velocity profile become uniform in shorter distance. In case (a), the x-velocity distribution becomes uniform after 25 cm from inlet section but this distance is shorter in the other cases.

Existence of inlet slot in the middle of the tank causes to formation non-zero vertical velocity. The positive and negative z-velocities causes to creation circulation zone.
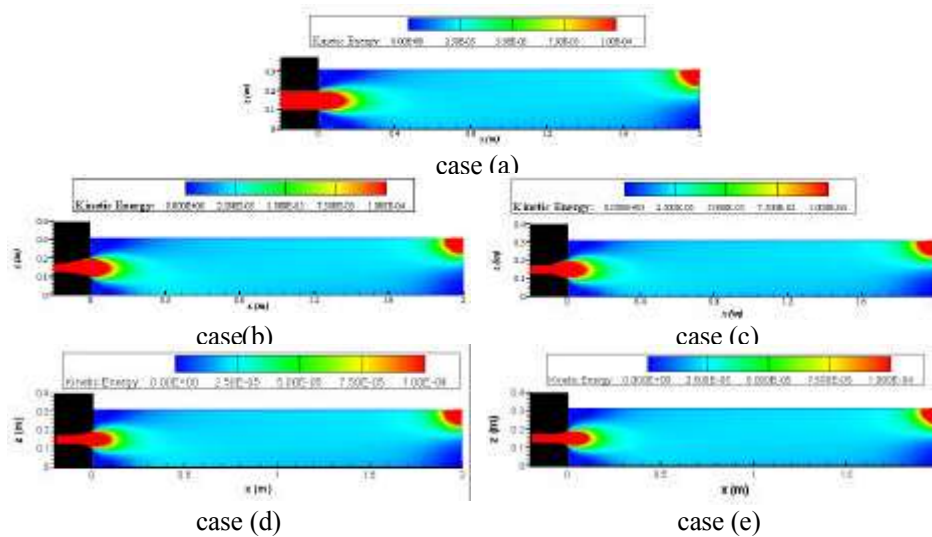
case (a)

case(b)

case (c)

case (d)

case (e)

Fig. 7: Turbulence kinetic energy

The circulation zone will be smaller if these vertical velocity decline in quantity and occupied space. Fig. 6 demonstrates the vertical velocity distribution in the first half of tank length. There is a small difference between a variety of cases and the changing of inlet form do not any considerable influence on distribution of vertical velocity in the inlet zone.

The additional function of inlet structure is reducing the kinetic energy (k) within the inlet zone of tanks. From Fig. 7, the turbulence kinetic energy is reduced in the tank from inlet zone to sedimentation zone. The modified cases could decrease the kinetic energy in the inlet zone. The high value of kinetic energy in the modified cases is smaller than rectangular shape and kinetic energy dissipated in the smaller area.

CONCLUSION

Energy dissipation is the main objective in designing a primary clarifier inlet. Different types of inlet aperture in rectangular primary clarifier were considered. These configurations are commonly intended to reduce high-velocity currents and avoid flow jets from travelling toward the effluent withdrawal area. Rectangular, steep and curved shapes of inlet slot are studied in this research. Uniform flow is generated in considerably shorter distance and the turbulence kinematic energy in these cases is slightly low. Moreover, the areas occupied by circulation zone in the modified cases are less than amount in the rectangular opening. As a result, expanding slot as inlet has significant effect in uniformity of flow from inlet zone to sedimentation zone and decrease turbulence kinematic energy and circulation zone.

REFERENCES

1. Krebs, P., D. Vischer and W. Gujer, 1995. Inlet-structure design for final clarifiers. Journal of Environmental Engineering, ASCE, 121(8): 558-564.
2. Das, D., T.M. Kenaith, D.S. Parker and E.J. Wahlberg, 1993. Floc Breakup in Activated Sludge Plants. Water Environment Research, 65(2): 138-145.
3. Imam, E., J.A. McCorquodale and J.K. Bewtra, 1983. Numerical Modeling of Sedimentation Tanks. Journal of Hydraulic Engineering, ASCE., 109(12): 1740-1754.
4. Fischerström, C.N.H., E. Isgard and I. Larsen, 1967. Settling of activated sludge in horizontal tanks. Journal of Sanitary Engineering Division, ASCE, 93(SA3): 73-83.
5. Tamayol, A., B. Firoozabadi and G. Ahmadi, 2008. Effects of Inlet Position and Baffle Configuration on Hydraulic Performance of Primary Settling Tanks. Journal of Hydraulic Engineering, ASCE., 134(7): 1004-1009.
6. López, P., Rodríguez, A.G. Lavín, M.M.M. López and J.L.B.d.l. Heras, 2008. Flow models for rectangular sedimentation tanks. Chemical Engineering and Processing, 47: 1705-1716.
7. Larsen, P., 1977. On the hydraulics of rectangular settling basins: experimental and theoretical studies. in, Lund Inst. of Technol., Lund, Sweden.
8. Goula, A.M., M. Kostoglou, T.D. Karapantsios and A.I. Zouboulis, 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.

9. Rostami, F., M. Shahrokhi, M.A. Md Said and R. Abdullah, Syafalni, 2011. Numerical modeling on inlet aperture effects on flow pattern in primary settling tanks. Applied Mathematical Modelling, 35: 3012-3020.

10. Yakhot, V. and L.M. Smith, 1992. The renormalization group, the-expansion and derivation of turbulence models. Journal of Scientific Computing, 7(1): 35-61.

11. Hirt, C., 1992. Volume-fraction techniques: Powerful tools for flow modeling. in: Flow Sci. Rep., pp: FSI-92-00.

12. Yakhot, V. and S.A. Orszag, 1986. Renormalization group analysis of turbulence. I. Basic theory. Journal of Scientific Computing, 1(1): 3-51.

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

14. Hirt, C.W., 1992. Identification and Treatment of Stiff Bubble Problems. in, Flow Science Inc.

15. FlowScience, 2009. Flow-3D user manual. in.

16. Imam, E., J.A. McCorquodale and J.K. Bewtra, 1983. Numerical Modeling of Sedimentation Tanks. J. Hydraulic Engineering, 109(12): 1740-1754.