ACTA TECHNICA JAURINENSIS
Vol. 15, No. 4, pp. 207-220, 2022
10.14513/actatechjaur.00672

Research Article

Numerical Investigation of the Effect Dimensions of Rectangular Sedimentation Tanks on Its Hydraulic Efficiency Using Flow-3D Software

Mohammad Javadi Rad[1,*], Fatemeh Rostami[2], Pedram Eshaghieh Firoozabadi[2]

[1]Department of Civil Engineering, University of Qom
Ghadir Blvd., Qom, 3716146611 Iran
[2]Department of Civil Engineering, North Tehran Branch, University of Islamic Azad
No 159, 7th Boostan St., Pasdaran Ave. North Tehran, Iran
*javadirad.mohammad@email.com

Submitted: 10/07/2022 Accepted: 14/11/2022 Published online: 18/11/2022

Abstract: Settling basins are among the essential units built to separate sediment suspended and within the inlet flow particles in water and wastewater treatment plants and irrigation canals. These basins' high efficiency requires proper design, creating a smooth and uniform flow along the basin, and reducing circulation zone as a factor in disrupting the sedimentation process. The present study investigates basin dimensions' effect on its flow pattern. Hence In the current study, the primary rectangular sedimentation basin was modelled three-dimensionally using Flow-3D software. This software takes advantage of two new advanced technique of VOF and FAVOR to model the free surface of the flow and the geometry, respectively. The dimensions of the basin were examined in two scenarios. In the first set-up, the length-to-width ratio was evaluated by increasing length and decreasing width simultaneously and the second part examined the length to depth ratio by decreasing depth and increasing width. In both situations, the volume and location of the inlet and outlet of the basin were constant and unchanged. The outcomes indicate that increasing the ratio of length-to-width and length to depth reduces the volume of the circulation zone significantly. The volume of these zones decreased from 53% for the L/W ratio of one (square basin) to 22% associated with the L/W ratio of eight. Likewise, the volume of these zones decreased by 38% as a result of increasing the L/d ratio from five to ten.

Keywords: Numerical Investigation; Rectangular Settling Tanks; Basins Dimensions; Length/Width; Length /depth.

I. INTRODUCTION

Wastewater treatment methods were originally developed for concerns about public health and environmental conditions. Since surface water is one of the important sources of water supply in different countries, research on sedimentation basins has a long history. Sedimentation tanks are responsible for the deposition of suspended particles using gravity. Because sedimentation tanks are part of the primary stages of treatment, their performance affects the efficiency of other treatment plant units. By entering the stream into the tank and reducing the flow velocity, it is possible to settle suspended particles denser than water [1]. Due to the high construction and maintenance costs of these basins, their optimal performance of them is crucial. Despite the importance of settling basins as one of the main units of treatment plants, the existing design methods rely heavily on the empirical formulas of researchers who need to consider the hydrodynamic details of flow and sediment fully.

Where the sedimentation tank is efficiently designed and operated, the suspended solids decrease by about two-thirds. Generally, these tanks are designed to provide a hydraulic retention time of about 1.5 to 2.5 hours based on the average flow rate [2]. Primary sedimentation tanks are designed based on the surface load, while secondary sedimentation basins are designed according to the surface and solid load. Influential factors in the sedimentation process include surface load, basin shape and depth.

Circulation zones are one of the most common adverse factors in sedimentation basins. These zones create by short-circuiting between the inlet and outlet of the tank, which resulted in dead zone development and optimal sedimentation reduction. Reducing the dead zones inside the tank provides a suitable potential for sedimentation. As the size of the sedimentation tanks increases, the loading capacity and, consequently the effective level of sedimentation increase. However, this requires large costs in its construction, operation, supply and maintenance. Therefore, the use of alternative methods to optimize performance and increase the efficiency of the sedimentation tank feat the existing conditions is necessary and inevitable.

Kawamura (2000) [3] concluded that many factors in water and wastewater treatment would affect the performance of the sedimentation tank. The four types of effects that were not considered until then are:

- Basin geometry, such as its structure, location and arrangement of inlet and outlet.
- Flow characteristics such as density effects may make the velocity profile uneven. This effect may cause a short circuit from the inlet to the outlet, re-sedimentation of sediment particles and increased turbulence intensity.
- Particle removal systems may disturb the flow's stability and uniformity.
- Environmental and climatic aspects, such as air, wind gap and internal temperature of the stream.

Razmi et al. (2008) [4] bridged the gap of experimental data and carried out a serious lab test using the Acoustic Doppler Velocimeter (ADV), which measures instantaneous speed in three directions. In addition to the non-baffle test, they examined the baffle's effect on the flow's hydrodynamics. Their results show that:

- The presence of a baffle in the primary zone is more effective than in the middle zone.
- The optimal location of the baffle is the point where the volume of the circulation zone of the fluid inside the basin is minimal.

Rostami et al. (2011) [5] simulated the flow characteristics in the primary sedimentation basin using a computational fluid dynamics model. In this modelling, an incompressible and non-floating liquid is assumed. Besides, a turbulent RNG model with Navier-Stokes equations is used. In order to investigate the hydraulic effects on the velocity profile, separation length, and kinetic energy, three different input positions and three baffles at the input have been simulated. In this model, the fluid flow moves through the network, and the free surfaces are tracked by the liquid volume (VOF) method. The effects of the number and locations of the input baffle in the flow field are shown. The results indicate that the input baffle's position affects the sedimentation basin's flow pattern. An increase in fissures and inlets can reduce kinetic energy in the input area and cause a uniform flow. Shahrokhi (2012) [6] investigated the reduction of kinetic energy in a 2-meter length laboratory sedimentation tank based on flow patterns. In this research, the tank's number and position of flow inputs have been investigated to increase productivity and efficiency. From the results, it has been observed that two inputs in the middle (relative to the height) and at the same distance (relative to the symmetry line of the basin) produce the highest efficiency according to the flow patterns.

Patziger (2016) [7] aimed to improve shallow circular secondary basins (SSTs) using numerical fluid dynamics (CFD) analysis and observed a direct relationship between inlet height and inlet jet radial length in shallow circular secondary basins. To calculate and evaluate the performance of the basin, Lee (2017) [8] investigated the effect of double perforated baffle on solids removal in rectangular secondary basins using CFD simulations. It was observed that the double-perforated baffle impedes longitudinal movement and reduces the density of suspended sediments in the wastewater. He also recommended the installation of this baffle in rectangular secondary basins to reduce the total operating costs of wastewater treatment plants. Liu (2017) [9] performed a numerical simulation of the effect of the baffle on the hydraulic properties of the flow and the particle removal efficiency from a sedimentation basin by combining two liquid and solid phases using $k-\varepsilon$ turbulence model. The length of their modeled basin was about 21 meters. Their results showed that with the horizontal distance of the baffle between 0.5 to 2.5 meters from the input, the amount of particle removal increases. Also, with increasing the submerged depth of the baffle from 0.5 to 2 meters and especially with increasing it from 0.5 meters to one meter, the amount of particle removal obviously increases.

Zanganeh et al. (2017) [10] in a laboratory model showed that the possibility of increasing the efficiency of primary sedimentation tanks by using thin layer plates will cause more sedimentation of suspended solids. Increasing the productivity of sedimentation tanks in the wastewater treatment plant is important for technical and economic reasons, so finding a model to increase the productivity of sedimentation tanks is necessary and essential. This study, which uses thin-film plates of minerals in the sedimentation tank, in the laboratory and using a mathematical model, actually compares the efficiency of the actual volume of the tank equipped using thin-layer plates in the laboratory model with the tank under normal conditions (without using thin layer plates). The results depict that the velocity in the tanks equipped with thin-layer plates of minerals was 1.6 m/s less than the tank in normal conditions.

Javadi Rad et al. (2017) [11] conducted research on the number of baffle structures in the actual sedimentation basin of the southern Tehran wastewater treatment plant. Based on the conformity of the numerical results with the experimental data, the $k-\varepsilon$ turbulence model was used to solve the flow turbulence, therefore the appropriate accuracy of the numerical results was assured. The optimal location of the first baffle was determined from the authors' previous research. To determine the optimal number of baffles, the results were compared between tank without baffles and optimal conditions of tank with one, two and three baffles. The outcomes reveal that the presence of baffle structure prevents the flow jet in the floor and hence increases the deposition efficiency of the initial sedimentation basins. In addition, by adding a baffle in the right position, the magnitude of the maximum speed, the volume of the circulation zone and the kinetic energy are reduced, so that by adding the first, second and third baffles the volume of the circulation zone decreases by 4.13, 4.44 and 4.63% respectively. Zhou et al. (2018) [12] adopted a three-dimensional model to evaluate the performance of a long rectangular secondary basin. The results of simulations showed that one baffle and two perforated baffles in a row have the best performance. The predicted results of the model are well matched with field observations.

Aminnejad and Lajevardi (2018), [13] investigated the effect of basin shape on the amount of sediment. For this purpose, he has considered sedimentation ponds with normal urban dimensions and conditions and has studied the flow field and sediment density in the basins using multiphase flows and numerical methods as well as Fluent software. Then, changes were made in its dimensions in terms of length, width and height with a fixed volume and its effect on efficiency was investigated. The results of calculations showed that contrary to previous researchers who considered the length of the basin as the most effective parameter in sedimentation, due to the coarseness of particles and geometric properties of urban basins, increasing the width of the pond in the direction perpendicular to the flow axis has the highest effect.

Alighardashi and Goodarzi (2019) [14] studied the hydraulic efficiency of sedimentation tanks. In that paper, they aim to investigate the effects of the depth and wind on the water's surface on the hydraulic efficiency of the sedimentation tanks in water and wastewater treatment plants. A verified two-dimensional numerical study was performed to evaluate the hydraulic performance of series settling tanks by four different depths of 2.5, 3, 3.5 and 4 (m) and wind velocities of 5 and 7 (m/s). In both rhetorical and typical situations, the Real Hydraulic Retention Time and the adequate volume of sedimentation tanks increase widely as the tank depth.

Bouisfi et al. (2019) [15] researched improving the removal efficiency of sedimentation tanks using different inlet and outlet positions using the Computational Fluid Dynamics method. Two different configurations have been proposed and tested with varying particle diameters and concentrations. The numerical model's ability to describe flow field behaviour inside the tank is confirmed by experimentation data. Results show that inlet and outlet position influence the flow field and removal efficiency of sedimentation tanks, especially in the case of fine particles (50 and 120μm).

Patel et al. (2021) [16] simulated a rectangular and circular primary clarifier. The modelling of clarifiers is carried out using one-dimensional flux theory or two-dimensional computational fluid dynamics. The experiments were performed for three operating conditions, i.e. high, medium and low solid concentrations, using a lab-scale setup. The one-dimensional sedimentation process model was carried out on the MATLAB platform, and simulation was carried out. Based on the simulation and experimental outcomes, the removal of suspended solids is found to be better in circular clarifiers compared to rectangular clarifiers for an influent flow rate of 134 mL/min with influent total suspended solids ranged from 300 to 600 mg/L.

In the present study, the length/width (L/W) and length/depth (L/d) ratio of the sedimentation basin dimensions is investigated. Due to the relatively large occupied area and the construction costs of rectangular basins, its dimensions are of great importance. In these basins, the flow is horizontally in the direction of the basin. The length-to-width ratio of these basins is usually between 2 and 5 and sometimes up to 7. The higher this ratio, the less likely it is that a short circuit will occur between the input and output. Monk [17] considered the L/W ratio of 6 to 7 to be appropriate due to the short circuit control. Research on the dimensions of the basin has rarely been done and most of them has been done to improve the performance of the basin by making baffles in the flow path. This indicates the need for research on the dimensions of tanks.

In past research, the ratios of length-to-depth and length-to-width were not investigated simultaneously, and the available data needed more for comparison. Also, a time gap was observed in the research related to the geometric survey of rectangular tanks.

The limitations of this research are associated with its laboratory dimensions and the fact that the flow is not affected by the concentration of materials. Of course, in primary sedimentation tanks, due to the concentration lower than 200 mg/litre (Stamo, 1989), the flow is rarely affected by the concentration of materials. Also, the input flow to the tank is assumed to be uniform [18].

An introduction to existing research and a review of past research were described. In the following sections, the equations governing the flow using the fluid volume of the fractional areas and the method used in the research, the preparation of the models and the description of the boundary conditions and the structure of the mesh and the verification of the numerical models and finally the discussion of the results and conclusions It is described.

Figure 1. *Rectangular settling basin*

II. Flow governing equations

The equations governing Newtonian incompressible fluid flow encompass the continuity equation and the momentum equations in three main directions. The most important law used in the movement of fluids and in the water flow is the law of conservation of mass. From this law and using the mass equilibrium relationship at the input and output for a very small fluid element, the continuity equation can result. The continuity equation for this element is as follows [19]:

$$\frac{\partial \rho}{\partial t} + \nabla.(\rho V) = 0 \tag{1}$$

Which results in:

$$V_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u A_x) + \frac{\partial}{\partial x}(\rho v A_y) + \frac{\partial}{\partial x}(\rho w A_z) = 0 \tag{2}$$

Where, ρ and V_f are flow density and deduction of open volume to flow respectively. u, v and w are also velocities of the flow in the principal directions x, y and z respectively.

After applying the continuity equation, the principle of motion size conservation or momentum in fluid motion must also be considered. The Navier–Stokes equations are the equations governing the motion of a viscous Newtonian flow with velocity components in the three principal directions x, y, and z as follows [6]:

$$\frac{\partial u}{\partial t} + \frac{1}{V_f}\left\{uA_x\frac{\partial u}{\partial x} + vA_y\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial \rho}{\partial x} + G_X + f_x \tag{3-a}$$

$$\frac{\partial v}{\partial t} + \frac{1}{V_f}\left\{uA_x\frac{\partial v}{\partial x} + vA_y\frac{\partial v}{\partial y} + wA_z\frac{\partial v}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial \rho}{\partial y} + G_y + f_y \tag{3-b}$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_f}\left\{uA_x\frac{\partial w}{\partial x} + vA_y\frac{\partial w}{\partial y} + wA_z\frac{\partial w}{\partial z}\right\} = -\frac{1}{\rho}\frac{\partial \rho}{\partial z} + G_z + f_z \tag{3-c}$$

Where G_X, G_y and G_z are acceleration of mass gravity and f_x, f_y and f_z are accelerations of viscosity.

III. Research Methods

Nowadays numerical methods are very popular since they are fast, inexpensive and also eliminate limitations of analytical and experimental methods. In numerical methods, differential equations are numerically integrated at different times and places. In summary, in numerical methods for solving differential equations, the networking range is first determined and using time and space discretization methods, the differential equations are transformed into simple and solvable algebraic equations. Computers solve the resulting numerical equations on all defined network points or finite volumes, the accuracy of which depends on the discretization method used. Due to the repetition of the solution in different time and space periods and the high volume of calculations, the accumulation of errors in these methods is possible.

Flow-3D software has been considered as one of the newest, most popular and powerful software for numerical solution of three-dimensional motion of fluids. Although this software has been used for modelling all fluids, due to its special features it is especially used in hydraulic problems and has provided highly acceptable answers in this field. Thanks to a powerful graphical user interface which makes Flow-3D a user-friendly software [20].

This software takes advantage of two new advanced technique of VOF and FAVOR to model the free surface of the flow and the geometry, respectively. These models include: Prantel mixing length, single equation transfer model, two equation model of $k-\varepsilon$, two equation RNG model, LES large vortex simulation model, which shows the possibility of modelling a wide range of turbulent flows using Flow–3D.

Flow-3D software is equipped with a volume of fluid (VOF) scheme to give a more realistic analysis of the free surface flow. Cartesian, staggered grids are employed to solve the RANS equations (Reynolds Average Navier–Stokes), composed of continuity and momentum equations. [21 ,22] The VOF method consists of three main components: the definition of the VOF function, a method to solve the VOF transport equation, and the setting of boundary conditions at the free surface. Within the frame of VOF methods, the interface is determined from the volume fraction F [21]. Here, F is the fraction function. In particular, F = 0 when a cell is empty and F = 1 when a cell is full. The free surface is located at a position on intermediate values of F (the user may usually define F = 0.5, but another intermediate value) [23].

IV. Setup model

The modelled sedimentation tank includes an inlet at a height of 10 cm from the bottom of the tank and an outflow at the end. The inlet and outlet of the flow in all models is constant in the total width of the basin and the volume of the tank. It should be noted that three-dimensional geometric shape of the tank was drawn in the Flow-3D software due to its simplicity. However, in case of complex geometry, it should be drawn in software such as 3D AutoCAD and the STL output should be taken then from it and entered into Flow-3D. In this research, $k-\varepsilon$ turbulence model is used. Due to the compliance of results from the $k-\varepsilon$ turbulence model and experimental data. In addition, the acceleration in the z direction was placed at -9.81.

The main characteristic of the Flow-3D software is using a rectangular meshing system with high changeability for networking. This feature makes the network and the model geometry separate from each other. In simpler terms, it does not use a geometry-connected mesh system [24]. The blocks must either be placed exactly next to each other, which is called the linked blocks, or they must be placed completely inside each other, which are called nested blocks. The mesh is defined independently for each of the three orthogonal coordinates. Thus, numerical simulations were conducted with various numbers of cells to find the grid-independent solution. In this modelling, 225, 102 and 55 cells were used for the model length, width and height of 2, 0.5 and 0.3 m respectively. The smallest and largest dimension of a cell is about 5 and more than 10 mm respectively. **Fig. 2** demonstrates a three-dimensional view of the settling tank as well as the model grid in Flow-3D. The FAVORize tool is used to grid the model. Before starting to run the model, this tool spectacles exactly how well the software has correctly identified the model geometry (**Fig. 3**). This will assist to increase or decrease the size or number of cells if necessary.

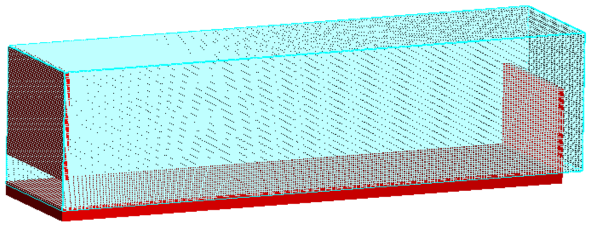


Figure 2. *A view of settling basin and its gridding drawn tree-dimensionally in Flow-3D software*

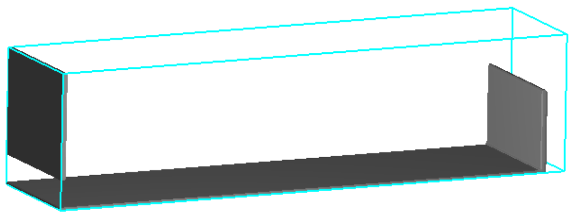


Figure 3. *Realized geometry of the model using FAVORize tool*

Input boundary condition: In order to determine unknown variables such as speed and pressure the input boundary condition should be defined in the model. Using the “Specified Velocity” option, the input velocity was defined as 0.04 m/s for all models.

Output boundary condition: By selecting the “Outflow” option for the output part after the overflow, the flow leaves the model horizontally.

Basin floor and wall boundary condition: Flow-3D with the power of the FAVOR method detects walls and floor, but the Wall option is selected for basin floor and walls. Due to the tank dimensions and the existence of a rigid boundary on both sides and its floor, the condition of zero velocity and no change of other parameters are applied by applying the boundary condition.

Boundary condition of flow-free surface: Flow-3D software uses the VOF method to solve free surfaces that simulate water and air boundary conditions as fluid volume. The boundary condition for the upper face in the vicinity of the air in the sediment tank is the "Symmetry" option; the symmetry condition is applied for zero gradient perpendicular to the boundary. The velocity and pressure changes in this boundary are zero to infinity. **Fig. 4** addresses the designated boundary conditions.

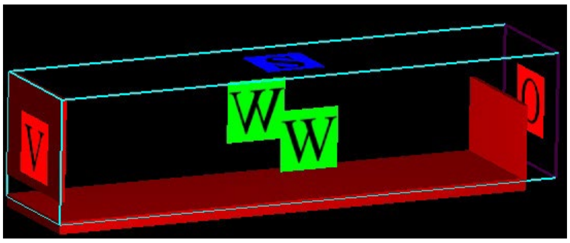


Figure 4. *A picture of the boundary conditions set for the study model*

In the current study the version of 11.2 of Flow-3D was used. All the numerical options were taken as the default value of the Flow-3D software. For example: The GMRES solver is a new algorithm in Flow-3D that was used in this numerical model. The GMRES pressure solver possesses good convergence, symmetry, and speed properties. However, it uses more memory than the SOR or the SADI methods.

The important topic in this research is the last two columns of **Table 2** including the length-to-width ratio and the length to depth ratio in a fixed volume of 0.3 m^3. Case No. 3, which is the same as the physical model constructed in Shahrokhi et al. lab, is the basic model of current study that is used in both scenarios of L/W and L/d ratio. To compare the L/W ratio, the results of cases 1, 2, 3 and 4 are used while for L/d ratio the outcomes of cases 3, 5 and 6 are evaluated.

V. VERIFICATION TEST

In order to verify the results of numerical modelling of the sedimentation tank, the numerical results were compared with the experimental data of Shahrokhi et al. The dimensions of their rectangular primary settling tank included a length (L) of 200 cm, a width (W) of 50 cm, a water depth (H) of 31 cm, an inlet opening height (H_{in}) of 10 cm, and an outlet weir height (H_w) of 30 cm.

The numerical results show good agreement with experimental data (**Fig. 5**), but some errors are observed near the bed, specifically in the regions near the inlet zone. The discrepancies between the result of the computational model and experimental measurements are probably due to the differences of the flow patterns in the inlet section. Also, **Table 1** shows the measured values root mean square errors (RMSE) of velocity profiles of experimental data and numerical results.

VI. RESULTS AND DISCUSSION

Based on the models addressed in **Table 2**, the results obtained by comparing the different dimensions of the rectangular basin are presented and reviewed. This section is described in two parts: L/W and L/d ratio. As previously mentioned, in all models, the volume of the tank is fixed and the inlet and outlet locations are unchanged. Besides, the inlet and outlet cover the entire width of the tank.

1. Comparison of L/W ratio of settling tank

In this section, considering fixed depth and volume equal to 0.3 m and 0.3 m^3, respectively, with simultaneous changes in the length and width of the tank, four different cases named 1, 2, 3 and 4 were raised whose dimensions are reported in **Table 2**.

Fig. 6 displays the streamlines and circulation zone for cases 1 to 4 in different length-to-width ratios. Cases 1 to 4 have L/W ratios of 1, 2, 4 and 8, respectively. It is observed that with increasing the L/W ratio (increasing the length and decreasing the width simultaneously) the volume of the circulation zone has decreased. In **Fig. 6 (a)** which is associated with the model with a length-to-width ratio of one (square tank), the volume of the circulation zone is calculated as equal to 53% of the total volume of the tank. While with the increase of length /width ratio, this volume has reached 22% in case number 4. The volume of circulation zone all four models is given in **Table 3**.

Fig. 7 depicts the kinetic energy contours for cases 1 to 4 with different length-to-width ratios. The length of the maximum kinetic energy (red zone) prevents proper particle sedimentation. This zone represents the flow jet after the inlet, therefore the shorter this zone the sedimentation performance the higher. Fig. 6-a shows that this zone covers about 80% of the length of the tank. By increasing the length-to-width ratio, the length of the zone with the maximum kinetic energy decreases, so that in case number 4, the length of this zone has reached less than 30% of the length of the sedimentation tank.

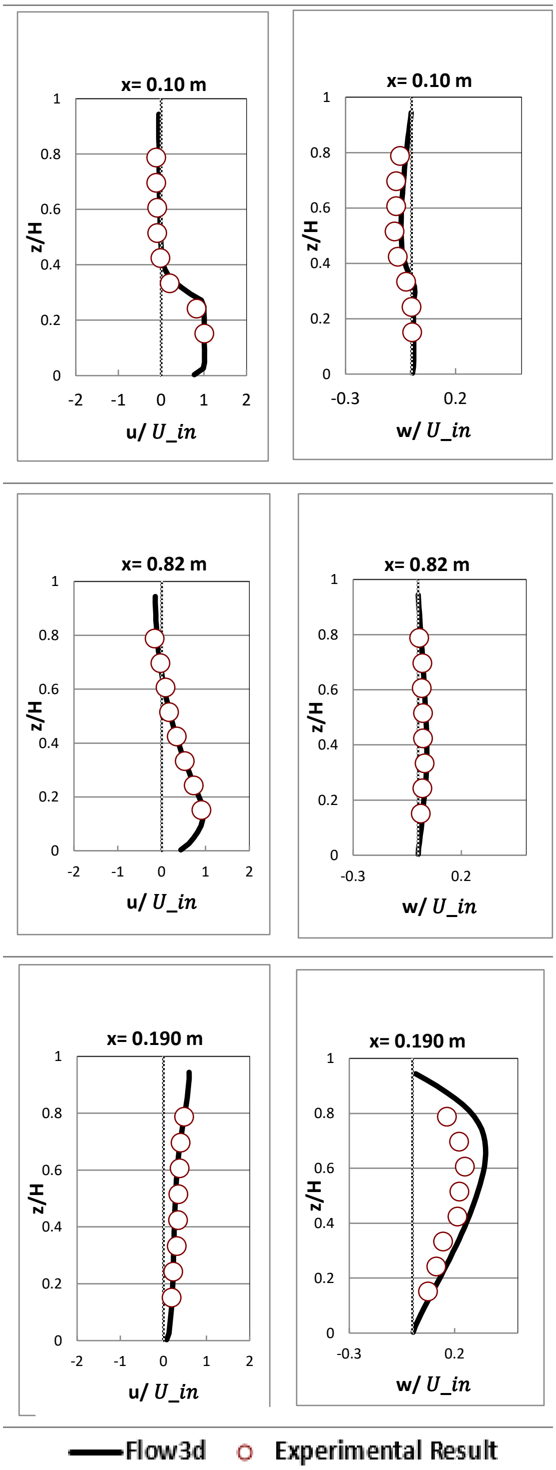


Figure 5. Comparison between experimental and computational x-velocity and z-velocity component

(z is transverse distance from the bed and u_{in} is inlet velocity)

***Table 1.** The measured values root mean square errors of velocity profiles of experimental data and numerical results*

Distance from the inlet (cm)	**10**	**82**	**190**
RMSE (in the x-direction)	0.11	0.07	0.08
RMSE (in the z-direction)	0.13	0.10	0.19

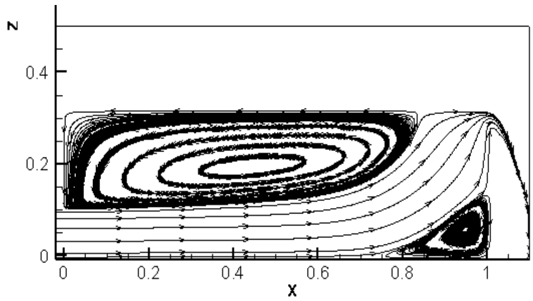
2. Comparison of basin L/d ratio

In the second part of the results, the ratio of L/d of the rectangular basin is evaluated. In this survey, the volume of the tank is fixed at $0.3\,m^3$ same as previous sub-section but this time the length of the tank has a fixed amount of 2 m and the width and depth are changing. Cases 5, 3 and 6 with the length-to-depth ratio of 5, 7 and 10, respectively are compared. Other specifications of the models are given in **Table 2**. Depth in the mentioned models is 0.4, 0.3 and 0.2 meters, respectively.

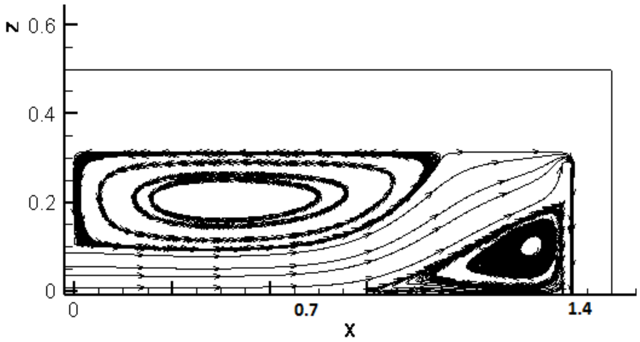
Fig. 8 demonstrates the streamlines from the inlet to the outlet within the basin. It is shown that by increasing the ratio of L/d (decreasing the basin depth), the volume of the circulation zones decreases. Circulation zones prevent sedimentation by creating transverse and sometimes reversible velocities. The minimum volume of the circulation zone is associated with case number 6 (with the maximum L/d ratio) by 16% of the total volume of the tank. Furthermore, an increase in the length-to-depth ratio results in a decline in the circulation zone length, which improves the performance of the tank. **Table 4** shows the circulation zones volume for models as a percentage of the total tank volume. However, case No. 6 with the highest length-to-depth ratio has the lowest circulation zone.

***Table 2.** Dimensions of simulated models of settling basin (all units are in meter)*

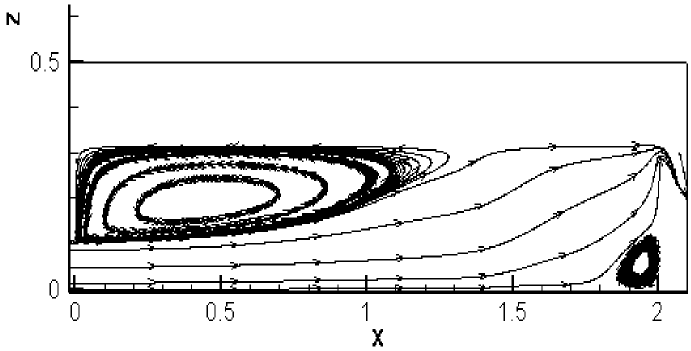
case number	*Length*	*Width*	*Depth*	*L/w*	*L/d*
1	1	1	0.3	1	7
2	1.4	0.700	0.3	2	7
3	2	0.500	0.3	4	7
4	4	0.250	0.3	8	7
5	2	0.375	0.4	4	5
6	2	0.750	0.2	4	10



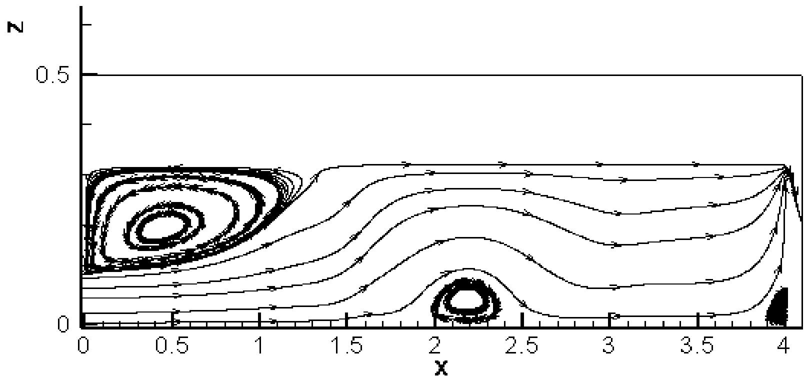
(a)



(b)



(c)



(d)

Figure 6. *Comparing the streamlines and circulation zone volume of rectangular basin for different L/W of: (a) case 1 (b) case 2 (c) case 3 (d) case 4*

Table 3. *Comparison of circulation zone volume for different L/W of rectangular basin.*

L/w	***1***	***2***	***4***	***8***
Circulation zone volume (%)	53	48	38	22

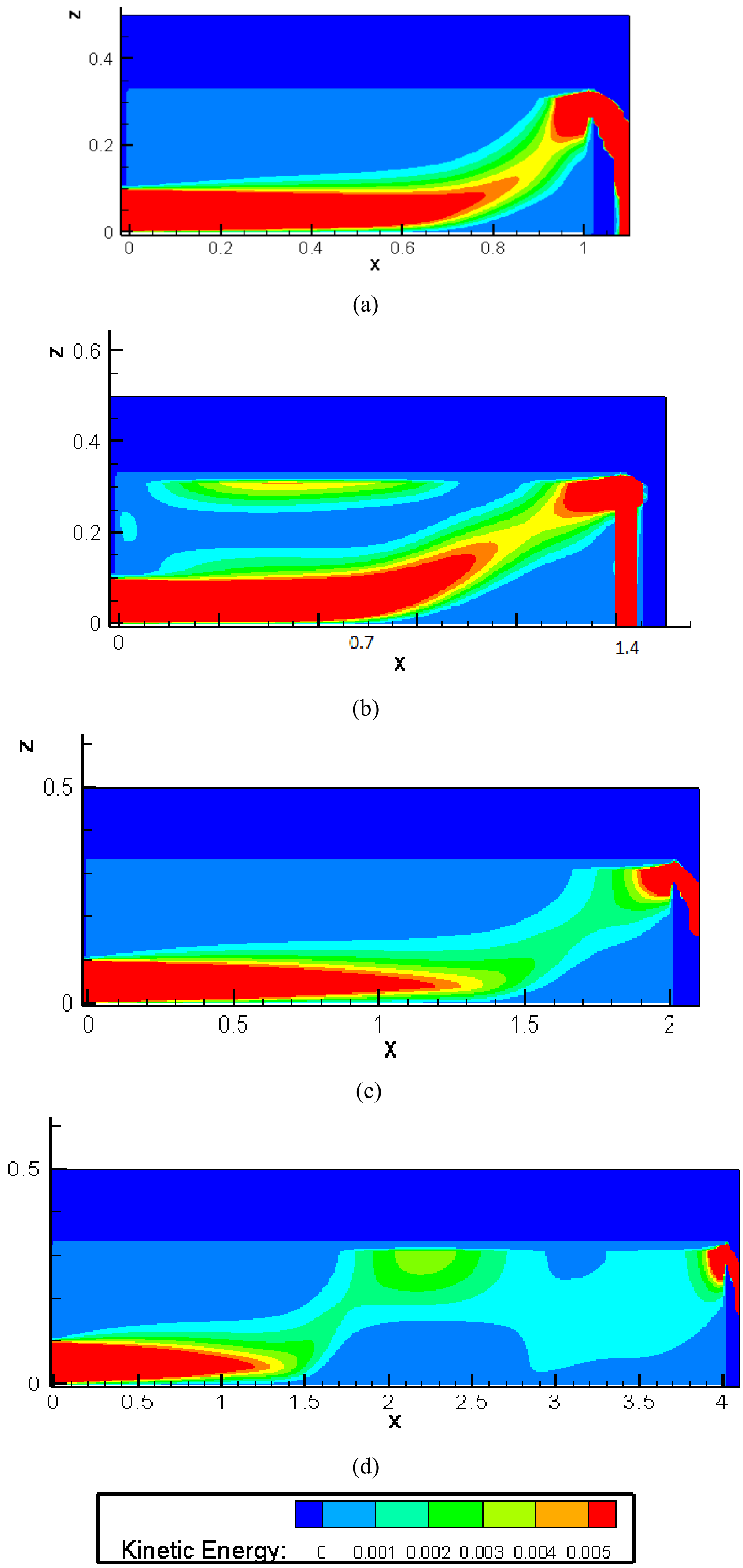
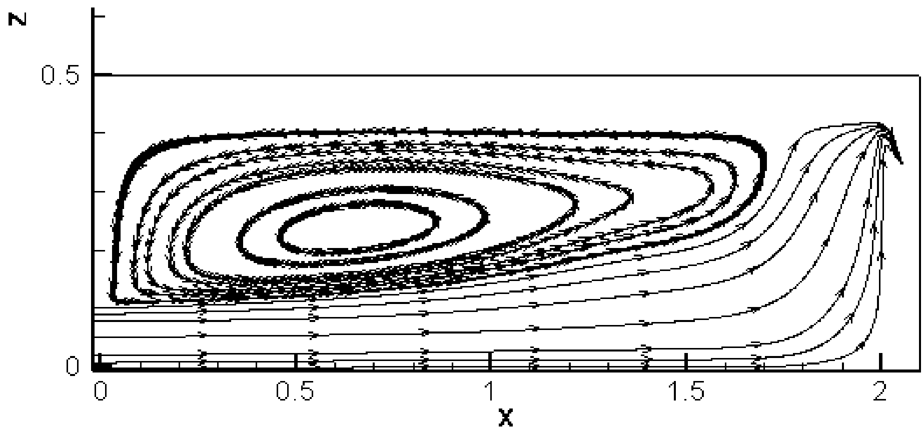
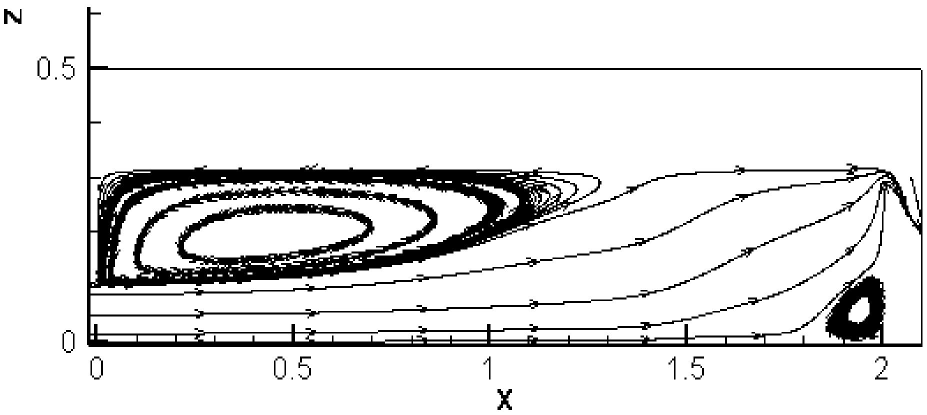


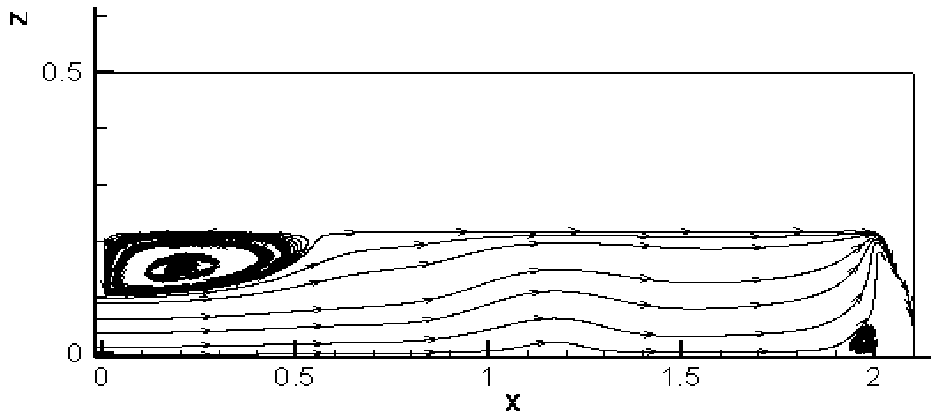
Figure 7. *Comparison of kinetic energy counters (m^2/s^2) for rectangular basin with L/W of: (a) case 1 (b) case 2 (c) case 3 (d) case 4*



(a)



(b)



(c)

Figure 8. *Comparison of streamlines and volume of circulation zones for a rectangular basin with the L/d ratio of: (a) case 5 (b) case 3 (c) case 6*

Table 4. *Circulation zone volume for different L/W of rectangular basin.*

L/W	***5***	***7***	***10***
Circulation zone volume (%)	54	38	16

Fig. 9 compares kinetic energy contours for different models of L/d ratio. The length of the area with maximum kinetic energy for cases 5, 3 and 6 is 1.5, 1.3 and 0.9, respectively, which includes 75, 65 and 45% of the total length of the basin, respectively. The results indicated that case No. 6 with the shortest

length of the maximum kinetic energy zone has the best performance in sedimentation. Also, in **Fig. 9-c**, associated with case No. 6, the maximum kinetic energy contours have moved upwards before the middle of the tank, which causes a smooth and uniform flow near the floor and consequently improves the efficiency of the tank.

VII. Concluding remarks

The required factor for the high efficiency of sedimentation basins is proper design and creation of a smooth and uniform flow along the basin which reduces circulation zones as an agent in disrupting the sedimentation process. In this research, the primary rectangular sedimentation basin was modelled using Flow-3D software in a three-dimensional environment. The purpose of this study was to investigate the effect of basin dimensions on the flow pattern within these basins. Hence, the present study was conducted in two scenarios. In both scenarios, the volume and location of the inlet and outlet of the basin were constant and unchanged.

The outcomes of this study reveal that by increasing the length-to-width ratio, the volume of the circulation zones decreases significantly. The volume of these areas increased from 53% for the L/W ratio of one (square basin) to 22% for the L/W ratio of 8. Besides, as the length of the basin increases and the width of the area decreases, the maximum kinetic energy also decreases. In such a way that by increasing the L/W ratio by 4 times this zone declined from 80% to about 30% of the total length of the tank. In the second part of the research, by increasing the L/d ratio of the basin from 5 to 10, the volume of circulation zones decreased from 54% of the total basin to 16%. Likewise, by increasing the L/d ratio at a depth of 0.2 m, the length of the zone with maximum energy reached 0.9 m, which is the lowest value compared to other models. In summary, for increasing the efficiency of sedimentation basins, it is suggested that the ratio of L/W and L/d adopt at least 4 and 7 respectively.

Other numerical models in the field of fluids dynamic can be used to compare with the results of this research. Also suggest to check more ratios of length to width and length to depth of basins in order to achieve more accurate dimensions. It is suggest to check the shape of the basins, for example, L-shaped tanks should be used in the research.

Author contributions

Mohammad Javadi Rad: Formal analysis, Visualization, Writing - original draft. Author: Conceptualization, Experiments, Theoretical analysis.

Fatemeh Rostami: Investigation, Resources, Visualization

Pedram Eshaghieh Firoozabadi: Supervision, Review and editing.

Disclosure statement

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

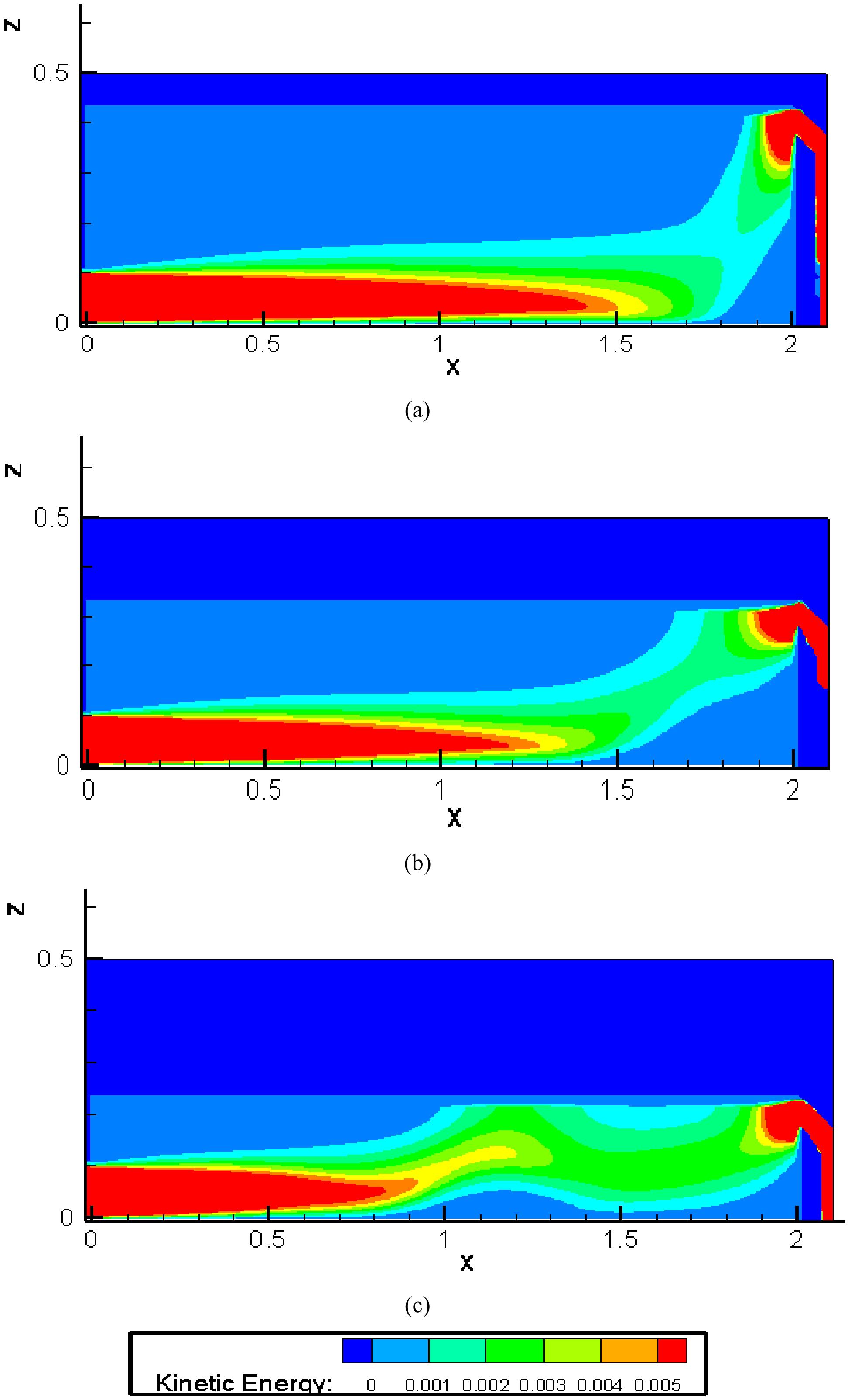


Figure 9. *Comparison of kinetic energy counters (m^2/s^2) for rectangular basin with L/W of: (a) case 5 (b) case 3 (c) case 6*

REFERENCES

[1] H. Asgharzadeh, B. Firoozabadi, H. Afshin, Experimental investigation of effects of baffle configurations on the performance of a secondary sedimentation tank, Sharif University of Technology (2011) pp. 938-949.

[2] Metcalf & Eddy 2003. Wastewater engineering treatment and reuse, New York, McGraw-Hill.

[3] S. Kawamura, Integrated design and operation of water treatment facilities, 2nd ed.; John Wiley and Sons Inc.: New York, NY, USA, (2000) pp. 159–160.

[4] A. Razmi1, B. Firoozabadi1, G. Ahmadi, Experimental and numerical approach to enlargement of performance of primary settling tanks, Journal of Applied Fluid Mechanics 2 (1) (2009) pp. 1-12.
https://doi.org/10.36884/JAFM.2.01.11850

[5] F. Rostami, M. Shahrokhi, Md Azlin Md Said, A. Rozi, Syafalni, Numerical modeling on inlet aperture effects on flow pattern in primary settling tanks, Applied Mathematical Modelling 35 (6) (2011) pp. 3012-3020.
https://doi.org/10.1016/j.apm.2010.12.007

[6] M. Shahrokhi, Influence of Baffle structure on the improvment efficiency of primary rectangular sedimentation tank, Universiti Sains Malaysia (2012) pp. 42-99.

[7] M. Patziger, Computational fluid dynamics investigation of shallow circular secondary settling tanks: inlet geometry and performance indicators, Chemical engineering research and design 112 (2016) pp. 122–131.
https://doi.org/10.1016/j.cherd.2016.06.018

[8] B. Lee, Evaluation of double perforated baffles installed in rectangular secondary clarifiers, (2017).

[9] Y. Liu, P. Zhang, W. Wei, Simulation of the effect of a baffle on the flow patterns and hydraulic efficiency in a sedimentation tank. Desalination and Water Treatment, 57 (54) (2016) pp.1-10.
https://doi.org/10.1080/19443994.2016.1157521

[10] H. Zanganeh, S. Narakorn, Three-dimensional VIV prediction model for a long flexible cylinder with axial dynamics and mean drag magnifications. Journal of Fluids and Structures (2016) pp. 127–146.
https://doi.org/10.1016/j.jfluidstructs.2016.07.004

[11] M. Javadi Rad, M. Shahrokhi, T. Rajaee, Numerical investigation of the number of baffles effect on the efficiency of primary sedimentation tank in true dimensions, Modares Civil Engineering Journal (M.C.E.J) 17 (5) (2017) pp. 49-59.

[12] S. Zhou, C. Vitasovic, J.A. McCorquodale, S. Lipke, M. DeNicola, P. Saurer, Improving performance of large rectangular secondary clarifier. Available online: (28 April 2017).
https://hydrosims.com/files/Optimization_Rectangular_Clarifiers.pdf

[13] B. Amin nezhad, A. Lajvardi, The effect of basin shape on the amount of sediment, Bibliography number: 4535528, (2017).

[14] A. Alighardashi, D. Goodarzi, Simulation of depth and wind effects on the hydraulic efficiency of sedimentation tanks, Water and Environment Journal, 34 (4) (2019) pp. 432-440.
https://doi.org/10.1111/wej.12478

[15] F. Bouisfi, A. Bouisfi, H. Ouarriche, M. E. Bouhali, M. Chaoui, Improving Removal Efficiency of Sedimentation Tanks Using Different Inlet and Outlet Position, FME Transactions, 47 (4) (2019) pp. 894-900.
https://doi.org/10.5937/fmet1904894B

[16] N. Patel, J. Ruparelia, J. Barve, Experimental and simulation study of rectangular and circular primary clarifier for wastewater treatment, Environmental Technology & Innovation, 23 (2021) pp.1-12.
https://doi.org/10.1016/j.eti.2021.101610

[17] R. Monk, J.F. Willis, Designing water treatment facilities. JounalAWWA, 79 (2) (1987) pp. 45-57.
https://doi.org/10.1002/j.1551-8833.1987.tb02798.x

[18] A. L. Stamou, E. W. Adams, W. Rodi, Numerical modeling of flow and settling in primary rectangular clarifiers, Journal of Hydraulic Research. 27 (1989) pp. 665-682.
https://doi.org/10.1080/00221688909499117

[19] C. W. Hirt, B. D. Nichols, Volume of Fluid (VOF) Method for the Dynamics of Free Boundaries. Journal of Computational Physics 39 (1) (1981) pp. 201-225.
https://doi.org/10.1016/0021-9991(81)90145-5

[20] Flow-3D Users Manuals, v 11.1. (2016).

[21] A. Ghaderi, M. Dasineh, F. Aristodemo, A. Ghahramanzadeh, Characteristics of free and submerged hydraulic jumps over different macroroughnesses, Journal of Hydro-informatics, 22 (6) (2020) pp. 1554-1572.
https://doi.org/10.2166/hydro.2020.298

[22] M. Ahmadi, A. Ghaderi, H. Mohammad Nezhad, A. Kuriqi, S. D. Francesco, Numerical Investigation of Hydraulics in a Vertical Slot Fishway with Upgraded Configurations, Water, 13 (19) (2021) pp. 1-23.
https://doi.org/10.3390/w13192711

[23] S. Abbasi, S. Fatemi, A. Ghaderi, S. D. Francesco, The Effect of Geometric Parameters of the Antivortex on a Triangular Labyrinth SideWeir, Water, 13 (1) (2020) pp. 2-25.
https://doi.org/10.3390/w13010014

[24] Flow-3D, Help, V.11.2, Flow Science Inc.