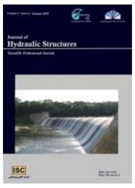
**Journal of Hydraulic Structures**
**J. Hydraul. Struct., 2023; 9(4): 76-87**
**DOI: 10.22055/jhs.2024.44817.1265**



# Effect of inlet and baffle position on the removal efficiency of sedimentation tank using Flow-3D software

Ali Poorkarimi[1]
Khaled Mafakheri[2]
Shahrzad Maleki[2]

## Abstract

Sedimentation due to gravitation is applied widely in water and wastewater treatment processes to remove suspended solids. This study outlines the effect of the inlet and baffle position on the removal efficiency of sedimentation tanks. Experiments were carried out based on the central composite design (CCD) methodology. Computational fluid dynamics (CFD) is used extensively to model and analyze complex issues related to hydraulic design, planning studies for future generating stations, civil maintenance, and supply efficiency. In this study, the effect of different conditions of inlet elevation, baffle's distance from the inlet, and baffle height were investigated. Analysis of the obtained data with a CCD approach illustrated that the reduced quadratic model can predict the suspended solids removal with a coefficient of determination of $R^2$ = 0.77. The results showed that the inappropriate position of the inlet and the baffle can have a negative effect on the efficiency of the sedimentation tank. The optimal values of inlet elevation, baffle distance, and baffle height were 0.87 m, 0.77 m, and 0.56 m respectively with 80.6% removal efficiency.

**Keywords:** Sedimentation tank, Particle removal, Central Composite Design, Computational Fluid Dynamics, Flow-3D

Received: 19 September 2023; Accepted: 02 March 2023

## 1. Introduction

Suspended solids in water can cause various adverse effects, including water pollution and a decrease in water quality. Gravity settling is a conventional method for separating suspended solids and excess materials in water and wastewater treatment processes. Sedimentation tanks, which are commonly used in water treatment plants, are designed for the removal of these suspended solids through the settling process. Due to the high construction cost of settling tanks, understanding the hydraulic flow in these tanks is important for optimal design. Existing design methods largely rely on empirical formulas and do not consider the hydrodynamic details of the system.

---

[1] Department of Civil Engineering, Faculty of Engineering, Fasa University, Fasa, Iran, Email: poorkarimi@fasau.ac.ir **(Corresponding author)**

[2] Department of Civil Engineering, Faculty of Engineering, Fasa University, Fasa, Iran.

The most suitable flow condition in settling tanks is when the flow is uniform. Due to the geometry of the settling tanks, an eddy current is generally formed inside these tanks. The presence of these vortex zones, also known as dead zones, reduces the effective volume of the tank. The vortex area creates a direct and short path from the inlet to the outlet, known as the short-circuiting phenomenon, which reduces the residence time of particles in the tank [1]. Moreover, these vortex zones not only prevent particles from settling in the tank but may also reimmerse settled materials back into the main flow of the tank [2]. Borna et al. (2014) using the finite volume method and comparing it with experimental results, demonstrated how numerical simulations can predict the distribution and sediment transport in settling tanks before sedimentation. They simulated three-dimensional flow in a rectangular tank and solved the Navier-Stokes equations using the finite volume method. Then, by comparing the velocity profiles in different sections of the tank, the settling efficiency was evaluated using the proposed model [3]. Rad et al. [4] investigated the effect of basin dimensions on its flow pattern by modeling a primary rectangular sedimentation basin three-dimensionally using Flow-3D software. The first scenario evaluated the length-to-width ratio, while the second scenario examined the length-to-depth ratio. In both situations, the volume and location of the inlet and outlet of the basin were constant. The outcomes showed that increasing the ratio of length-to-width and length-to-depth significantly reduced the volume of the circulation zone [4].

After the advent of computational fluid dynamics (CFD), there was rapid development in the design and optimization of sedimentation tanks [5]. Various experimental and computational studies have been conducted to enhance the performance of settling tanks, suggesting various methods including the use of baffles in the tank floor, changing the inlet position, creating a slope in the tank floor, and altering the depth of the tank [6-8]. Based on the study by Gharagozian [9], among the parameters affecting the efficiency of sedimentation ponds, the two parameters of slope and depth have the least impact. On the other hand, the use of baffles in the tank floor has always been an effective solution in increasing settling efficiency. The height and location of the baffles are significant factors in determining the quality of the effluent water [9]. Shohrokhi et al. [1] found through a laboratory-scale study that the presence of a vortex area reduces the performance of settling tanks. Installing a baffle structure in a suitable position can reduce the occurrence of these zones. They demonstrated that a higher number of baffles decreases horizontal velocity and provides a greater chance for the settling of suspended particles [2, 10]. Razmi et al. [11] demonstrated that a baffle structure can increase the efficiency of settling tanks. The results of their experiments showed that installing a baffle near the inlet and at a height close to 25-30% of the water depth improves the efficiency of the settling tank [11].

In a study conducted by Liu et al. [12], a combination of liquid and solid phases and the k-ε turbulence model were used. The effect of a baffle on the removal efficiency of suspended solids in a 21 m-long settling tank was modeled. The results indicated that the particle removal increased with the horizontal distance of the baffle between 0.5 to 2.5 m from the inlet. Moreover, an evident increase in particle removal was observed with an increase in the immersion depth of the baffle from 0.5 m to 2 m [12]. Saeedi et al. [13] studied the hydraulic characteristics of flow with different baffle configurations on the basin floor and presented the best baffle geometry. The results indicated that baffles installed closer to the basin inlet with lower height, and those installed further from the basin inlet with higher heights had the greatest impact on sedimentation efficiency [13]. In the study conducted by Miri et al. [14], the finite volume method was used for the 3D simulation of flow in a rectangular basin using continuity and Navier-Stokes equations. Diffusion and transfer of sediment concentration were solved simultaneously with hydraulic flow equations to evaluate the sedimentation style in the pre-settled pool. Finally, two different states of the baffle located in the pond inlet including one-sided and two-sided baffles were investigated and the results were compared with the state without the baffle. The numerical results of sediment concentration profiles at different depths of the pond were well consistent with the laboratory results [14].

Moreover, the angle of the installed baffle has been investigated by various studies. Heidari et al. [15] conducted a series of experiments to investigate the effect of a baffle and its angle on the removal efficiency of suspended solids in an 8 m-long straight channel. They performed numerical analysis using Fluent software. The experimental and numerical results were consistent with each other, and the optimal installation angle was determined to be approximately 60 degrees [15]. Guo et al. [16] used the COMSOL computational fluid dynamics code to investigate the removal of fine particles in a tertiary sedimentation tank equipped with an adjustable baffle. The finite element method was used to consecutively solve the equations of continuity and momentum for solid-fluid motion. Results indicated that installing an adjustable baffle at different angles (i.e., 30°, 45°, and 60°) in the sedimentation tank resulted in a lower overflow rate and mixing intensity compared to not having one, leading to improved particle removal efficiency. However, the sedimentation tank with an adjustable baffle positioned at a 30° angle provided the highest settling efficiency [16].

Although numerous studies have been conducted on settling tanks, most of them have followed the conventional approach of changing one factor at a time (OFAT) while keeping the others constant. It is widely recognized that the response surface methodology (RSM) based on the central composite design (CCD) offers more advantages for the design of experiments (DOE) analysis and modeling. This approach requires less time, resources, and experimental work compared to the conventional OFAT method for modeling and optimizing processes. By employing RSM, it becomes possible to estimate the linear, interaction, and quadratic effects of the factors and develop a prediction model for the response variable.

According to the literature, the inlet and baffle position are two important parameters affecting the performance of a settling tank, however, the combined effects of them have not been studied. Therefore, in this study, three factors of inlet elevation, baffle height, and distance of the baffle from the inlet are studied as effective parameters. For that, CCD was used to simultaneously investigate the interactions of these factors on the performance of the settling tanks. Subsequently, simulations were conducted using the Flow-3D software to analyze different scenarios and generate response surfaces. Finally, by analyzing various scenarios and plotting response surfaces, the optimal condition was determined.

## 2. Methodology

### 2.1. Design of rectangular tank

To model the flow in a rectangular settling tank, it was assumed that the inflow rate into the tank is 120 L/s. Considering a retention time of 60 s, and a length-to-width ratio of 6, the dimensions of the tank were determined to be 6×1×1.2 m as shown in Figure 1.

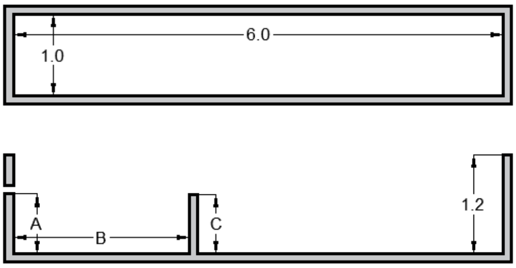


**Figure 1. Plan and longitudinal section of the tank.**

## 2.2. Inflow water specification

The water temperature for simulation was considered as 20 ºC. The concentration of suspended solids in the influent water was assumed to be 200 mg/L, and the density of particles was taken as 2650 kg/m$^3$. The particle size distribution and mass distribution were considered according to Table 1. The grading curve for these particles is shown in Figure 2.

**Table 1. Specifications of suspended solids entering into the tank.**

| Particle number | Particle diameter (mm) | Distribution of particle size (%) | Particle mass distribution (%) |
|---|---|---|---|
| 1 | 0.11 | 1.5 | 0.003 |
| 2 | 0.115 | 2.5 | 0.005 |
| 3 | 0.12 | 5 | 0.01 |
| 4 | 0.13 | 8.5 | 0.017 |
| 5 | 0.14 | 15.5 | 0.031 |
| 6 | 0.15 | 20.5 | 0.041 |
| 7 | 0.175 | 18.5 | 0.037 |
| 8 | 0.19 | 13 | 0.026 |
| 9 | 0.21 | 9 | 0.018 |
| 10 | 0.25 | 6 | 0.012 |

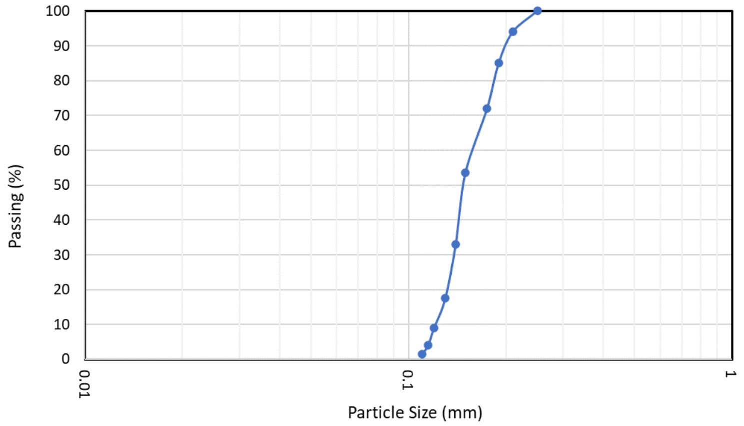


**Figure 2. Grading curve for the suspended solids.**

## 2.3. Mesh independence study

Determining the appropriate number of grids for simulation using CFD software is crucial in conducting accurate and time-efficient analyses. In this study, the Flow-3D software was utilized for simulation purposes. This software employs Reynold's Averaged Navier-Stokes (RANS) equation and a structured tetragonal mesh. To assess mesh independence, simulations were conducted with varying grid sizes, and the velocity values were measured at eight points along the X-axis using the Prob tool. The results showed that for meshes with sizes of 2 cm or smaller, there was a negligible difference in the measured velocity values. Therefore, to increase speed and reduce computation time, a mesh with a size of 2 cm was chosen for simulation and the number of mesh elements reached 2,120,625. The results related to mesh independence from the network are summarized in Table 2.

**Table 2. Investigation of mesh independence**

| Mesh size (cm) | Number of grids | Velocity magnitude (m/s) | | | | | | | |
|---|---|---|---|---|---|---|---|---|---|
| | | Prob1 | Prob2 | Prob3 | Prob4 | Prob5 | Prob6 | Prob7 | Prob8 |
| 10 | 16,575 | -0.170 | -0.166 | -0.050 | -0.044 | 0.152 | 0.206 | 0.058 | 0.028 |
| 7.5 | 40,200 | -0.12 | -0.10 | -0.03 | 0.00 | 0.09 | 0.00 | 0.03 | 0.03 |
| 5 | 136,500 | -0.104 | -0.090 | -0.078 | -0.021 | 0.032 | 0.108 | 0.005 | 0.014 |
| 2.5 | 1,062,000 | -0.060 | -0.087 | -0.074 | -0.022 | 0.020 | 0.126 | 0.001 | 0.007 |
| 2 | 2,120,625 | -0.05 | -0.08 | -0.08 | -0.03 | 0.13 | 0.11 | 0.08 | 0.06 |
| 1.75 | 3,190,600 | -0.05 | -0.08 | -0.08 | -0.03 | 0.12 | 0.11 | 0.08 | 0.07 |
| 1.5 | 5,066,100 | -0.05 | -0.08 | -0.08 | -0.03 | 0.12 | 0.08 | 0.08 | 0.07 |

## 2.4. Design of experiments

Application of the DOE procedure seems to be among the best methodologies for studying and improving complicated processes. The RSM is a statistical technique that can determine the influences of individual variables and their interactions on a response. In this study, CCD was applied for experimental design and evaluation of the effects of different independent variables and their interactions to find optimal operating conditions for the sedimentation of suspended solids. Design Expert version 13.0 was used for experimental design and graphical analysis of the obtained data.

The removal of suspended solids is affected by temperature, viscosity, solids density, horizontal velocity, and design parameters. As indicated in Table 3, three independent variables of inlet elevation, horizontal distance from inlet to baffle, and baffle height were coded as A, B, and C, respectively, according to Equation (1):

$$x_i = \frac{X_i - X_0}{\delta X} \tag{1}$$

where $x_i$ is the coded value of the *i*th independent variable, $X_0$ is the value of $X_i$ at the center point, and $\delta X$ presents the step change value. Having conducted some pretests for each variable, a proper range was selected, and this range was divided into proper intervals. The response of this design is suspended solids removal efficiency (SSRE%), which was calculated by Equation (2):

$$SSRE\,(\%) = \frac{C_0 - C_i}{C_0} \times 100 \tag{2}$$

where $C_0$ is the concentration of the suspended solids in the inflow water and $C_i$ is the concentration of the suspended solids in the outlet. The total number of experiments based on CCD was 15, including 8 factorial points, 6 axial points based on α = 2, and 1 center point.

**Table 3. Effective parameters and their selected values in five levels.**

| Effective parameters | symbol | Unit | Values and levels | | | | |
|---|---|---|---|---|---|---|---|
| | | | -2 | -1 | 0 | 1 | 2 |
| Inlet elevation (m) | *A* | m | 0.125 | 0.375 | 0.625 | 0.875 | 1.125 |
| Baffle distance (m) | *B* | m | 0.5 | 1.75 | 3 | 4.25 | 5.5 |
| Baffle height (m) | *C* | m | 0 | 0.25 | 0.5 | 0.75 | 1 |

## 3. Results and Discussion

### 3.1. CFD modeling

After designing the experiments (Table 4), the sedimentation tank geometry, which was prepared in AutoCAD, was introduced to the Flow-3D software. The fluid and suspended particle specifications were then given as input to the software. The concentration of suspended solids at the inflow was considered to be 200 mg/L, based on the particle size distribution described in Table 1. The Baffle Tool in the Flow-3D was used to measure the concentration at the outflow. The results of velocity measurement for 15 Runs, following the experimental design, are presented in Figure 3. Run 13 corresponds to a case when the baffle was not used in the tank.

The percentage removal for particles with different diameters in each run is shown in Figure 4. As observed, smaller diameter particles have lower removal efficiencies in all runs, to the extent that particles with a diameter of approximately 0.1 mm escape from the tank. However, particles with a diameter of approximately 0.2 mm and larger are almost completely removed.

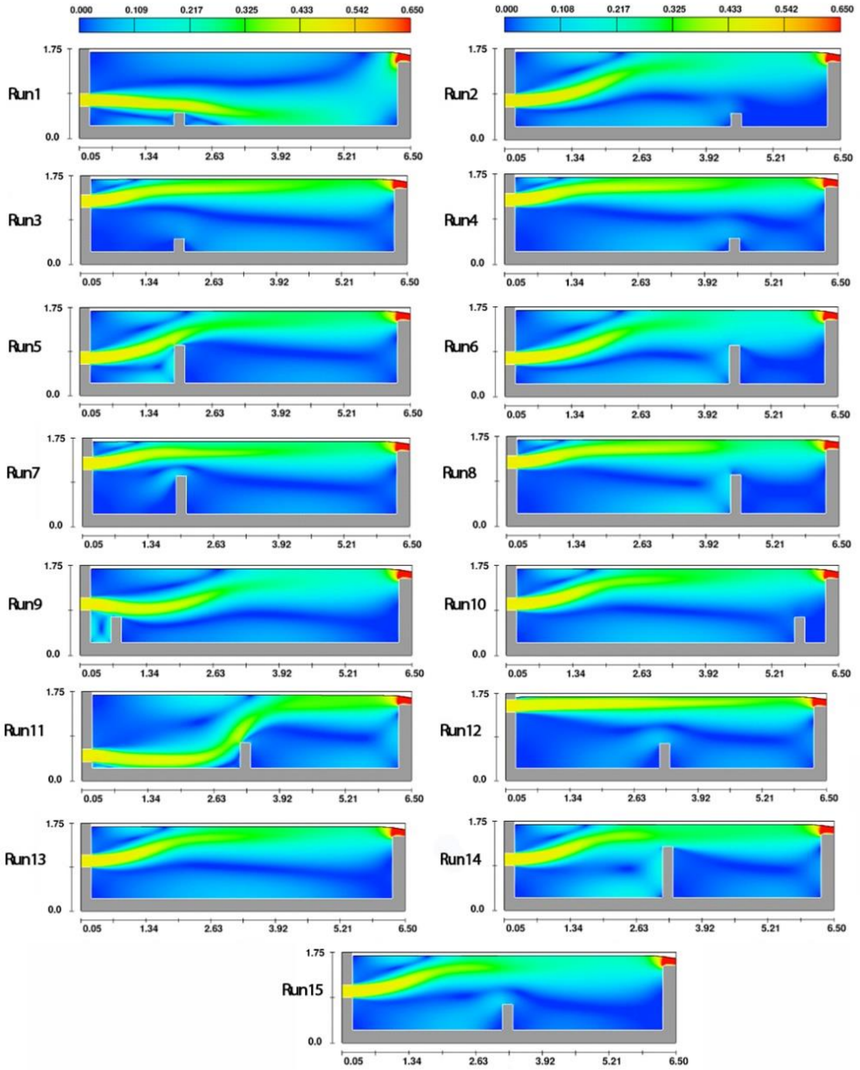


Figure 3. Computed contour of velocity magnitude (m/s) for Run 1 to Run 15.

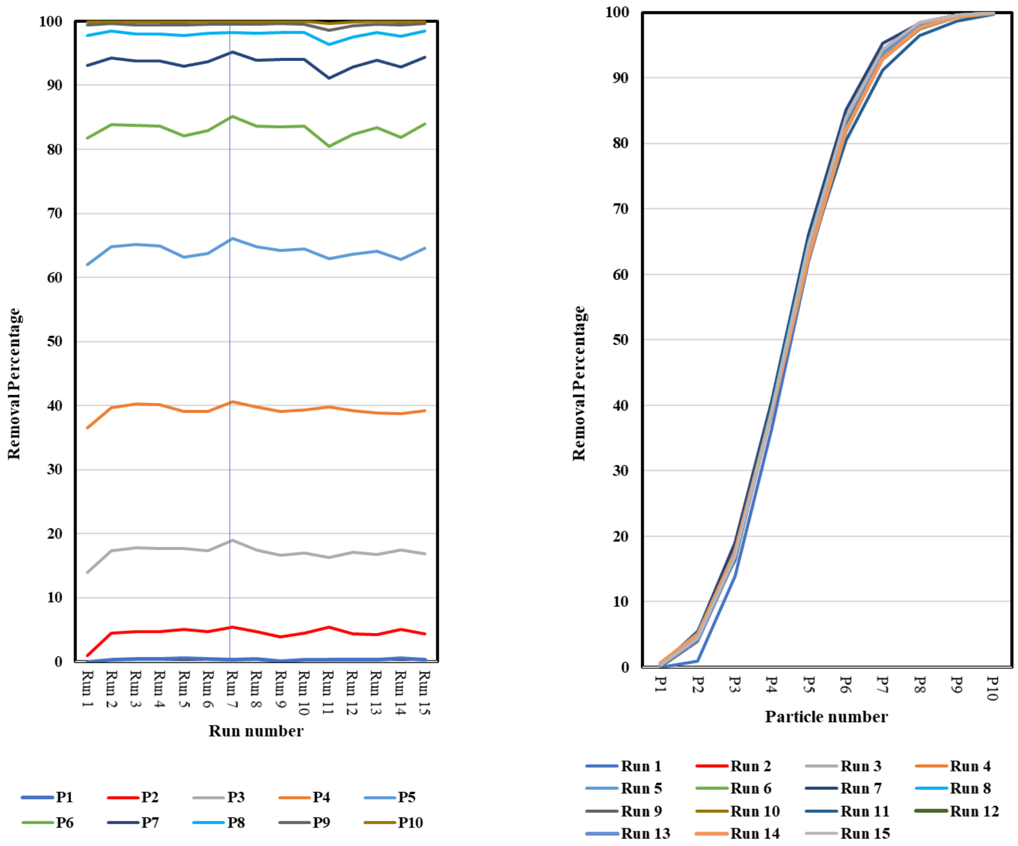


**Figure 4. Percentage removal of particles with different diameters in all models.**

## 3.2. Modeling by RSM

The measured removal percentage of particles with CFD along with the predicted values from Design Expert software for 15 designed runs are shown in Table 4. In Run 13, which corresponds to the case where the baffle was not used, the measured removal efficiency of suspended particles is 75.9%. The lowest removal efficiency is observed in Run 1, which is equivalent to 69.3%. In this case, the inlet elevation is 0.375 m, the baffle distance from the inlet is 1.75 m, and the baffle height is 0.25 m. These results indicate that if the baffle is not chosen correctly, it can have a negative impact on the removal efficiency. The highest measured removal efficiency is observed in Run 7, which is equivalent to 79.5%. In this case, the inlet elevation is 0.875 m, the baffle height is 0.75 m, and the baffle distance from the inlet is 1.75 m.

**Table 4. Removal percentage of suspended particles in different runs.**

| Run | *A*:Inlet elevation (m) | *B*: Baffle distance (m) | *C*: Baffle height (m) | $C_{in}$ (mg/L) | $C_{out}$ (mg/L) | *SSRE (%)* | |
|---|---|---|---|---|---|---|---|
| | | | | | | Measured | Predicted |
| 1 | 0.375 | 1.75 | 0.25 | 200 | 61.5 | 69.3 | 79.2 |
| 2 | 0.375 | 4.25 | 0.25 | 200 | 41.82 | 79.1 | 79.4 |
| 3 | 0.875 | 1.75 | 0.25 | 200 | 41.98 | 79.0 | 77.5 |
| 4 | 0.875 | 4.25 | 0.25 | 200 | 42.8 | 78.6 | 75.7 |
| 5 | 0.375 | 1.75 | 0.75 | 200 | 51.96 | 74.0 | 78.6 |
| 6 | 0.375 | 4.25 | 0.75 | 200 | 47.36 | 76.3 | 78.8 |
| 7 | 0.875 | 1.75 | 0.75 | 200 | 41.04 | 79.5 | 77 |
| 8 | 0.875 | 4.25 | 0.75 | 200 | 43.04 | 78.5 | 75.1 |
| 9 | 0.625 | 0.5 | 0.5 | 200 | 44.96 | 77.5 | 78.6 |
| 10 | 0.625 | 5.5 | 0.5 | 200 | 43.94 | 78.0 | 77 |
| 11 | 0.125 | 3 | 0.5 | 200 | 59.36 | 70.3 | 80.1 |
| 12 | 1.125 | 3 | 0.5 | 200 | 51.56 | 74.2 | 74.7 |
| 13 | 0.625 | 3 | 0 | 200 | 46.06 | 75.9 | 78.2 |
| 14 | 0.625 | 3 | 1 | 200 | 53.74 | 73.1 | 77 |
| 15 | 0.625 | 3 | 0.5 | 200 | 42.44 | 78.8 | 77.8 |

Based on the results obtained by software, a quadratic equation is suitable for the removal efficiency of the suspended particles. Equation (3) represents the reduced quadratic regression model for the measured values.

$$Efficiency = 78.83 + 1.54A + 0.731B - 0.206C - 1.69 * AB - 1.5A^2 - 0.94C^2 \qquad (3)$$

Regression coefficients in Equation (3) were determined using analysis of variance. The coefficients represent intensity, with signs indicating the positive or negative influence of each variable on the response. It is evident from Equation (3) that the baffle height has a net negative impact on the removal efficiency of the settling tank, while the inlet elevation and the baffle distance have positive impacts. Moreover, the interaction of the inlet elevation and the baffle distance affects the response of the removal efficiency. According to Table 5, The *p*-value and *F*-value of the model are 0.027 and 4.6, respectively, indicating that the applied model is meaningful and significant.

**Table 5. ANOVA for the response surface reduced quadratic model for particle removal efficiency.**

| Source | Sum of Squares | DF | Mean Square | *F*-value | *p*-value | |
|---|---|---|---|---|---|---|
| Model | 116.6 | 6 | 19.43 | 4.55 | 0.0266 | significant |
| A-Inlet Elevation | 38.13 | 1 | 38.13 | 8.92 | 0.0174 | |
| B-Baffle Distance | 8.56 | 1 | 8.56 | 2 | 0.1949 | |
| C-Baffle Height | 0.6806 | 1 | 0.6806 | 0.1592 | 0.7003 | |
| AB | 22.78 | 1 | 22.78 | 5.33 | 0.0498 | |
| $A^2$ | 43.7 | 1 | 43.7 | 10.22 | 0.0127 | |
| $C^2$ | 17.11 | 1 | 17.11 | 4 | 0.0804 | |
| Residual | 34.2 | 8 | 4.27 | | | |
| Cor Total | 150.8 | 14 | | | | |

The relationship between the predicted and measured values of particle removal efficiency is shown in Figure 5. The measured values of the percentage removal were obtained by CFD modeling, while predicted values were generated using Equation (3). Figure 5 demonstrates that predicted values are in good agreement with the measured values ($R^2 = 0.77$).

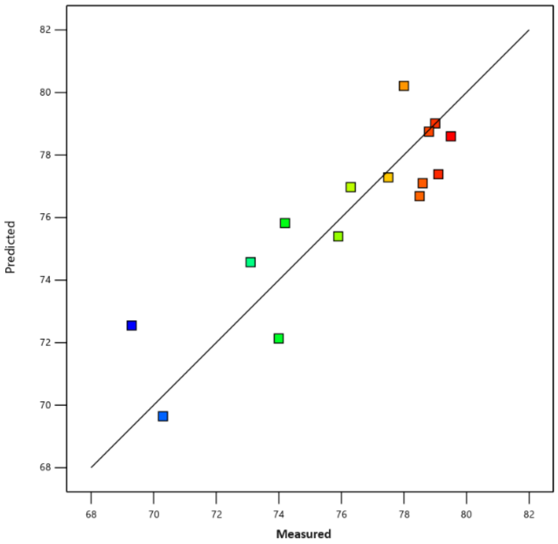


**Figure 5. The predicted versus measured responses for particle removal efficiency.**

The individual effect of each factor on the removal efficiency is shown in Figure 6. As observed, the distance of the baffle from the inlet has a linear effect on the efficiency of suspended particle removal. The height of the baffle and the elevation of the inlet have optimal values. By increasing the values of baffle height and inlet elevation, the removal efficiencies first increase and then decrease.

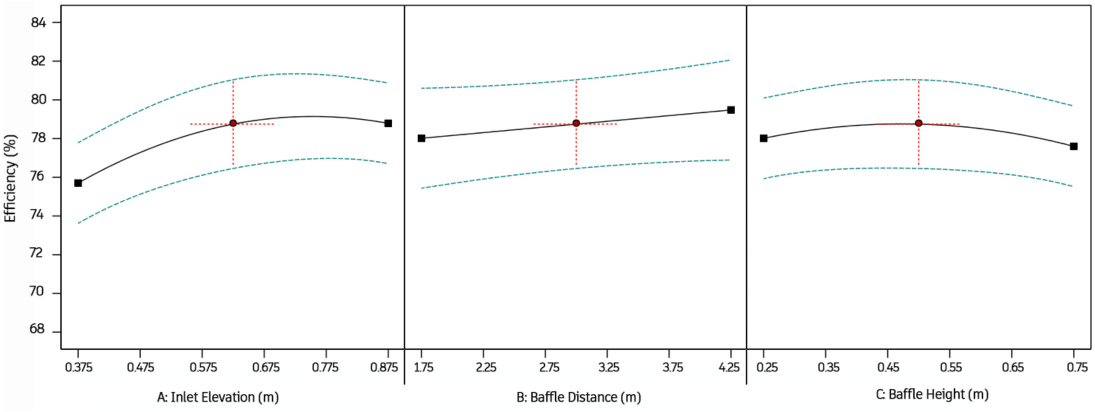


**Figure 6. Effect of each factor on removal efficiency.**

The optimization of suspended solids removal efficiency was carried out by variation of influential parameters of inlet elevation (0.125-1.125 m), baffle distance (0.5-5.5 m), and baffle height (0-1 m). The optimum condition for removal efficiency was reached by inlet elevation, baffle distance, and baffle height of 0.87 m, 0.77 m, and 0.56 m respectively. Under this condition, removal efficiency was predicted by the model, which is presented in Table 6.

**Table 6. Optimum values obtained for effective parameters and response.**

| Inlet elevation (m) | Baffle distance (m) | Baffle height (m) | Removal efficiency (%) | |
|---|---|---|---|---|
| | | | Predicted | Actual |
| 0.87 | 0.77 | 0.56 | 80.6 | 79.6 |

To ensure the correctness of the optimal conditions extracted from the model, a three-dimensional geometry plan with optimal conditions was made and run again. Figure 7 shows the computed contour of velocity magnitude in the optimal conditions. The percentage of particle removal, in this case, was 79.6%, which has a good correlation with the model optimization results (Table 6).

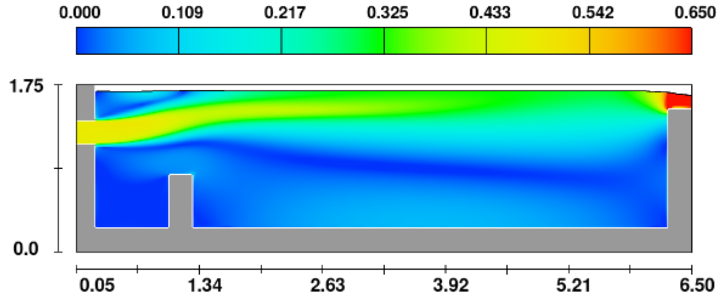


**Figure 7. Computed contour of velocity magnitude (m/s) for optimal conditions.**

In a study done by Razmi et al. [11], it has been shown that when the height of the baffle is 25-30% of the sedimentation tank's depth, the efficiency is maximum. In the current study, this issue was confirmed and showed that the efficiency has an extreme value compared to the height of the baffle. Also, regarding the baffle distance from the inlet, the optimal condition was achieved when the baffle was close to the inlet zone. This issue has been confirmed in a study by Saeedi et, al. [13] and it has been shown that in certain conditions when the baffle is closer to the inlet, the efficiency of the sedimentation basin is higher.

## 4. Conclusions

Sedimentation tanks are critical to water and wastewater treatment plants and must operate at their maximum potential. It is common for sedimentation tanks to be overdesigned due to a lack of knowledge about hydraulics, resulting in unnecessary expenses and excessive water wastage. To ensure optimal settling, a uniform flow field should be created in the tank. However, creating a circulation zone can disrupt the flow uniformity and negatively impact tank efficiency.

This study utilized numerical approaches to examine the effects of the inlet's elevation, baffle's position, and baffle's height on the flow field and performance of the settling tanks. A central composite design was employed to investigate the individual and combined effects of these parameters. For that, Flow-3D was used for the simulation of each experiment based on CCD to generate the response surface. The quadratic regression model was created to predict the settling performance of the settling tank. According to the analysis of variance, the elevation of the inlet and the distance of the baffle from the inlet have positive effects on the removal efficiency while the height of the baffle has a net negative effect. Moreover, there is an interaction between the elevation of the inlet and the distance of the baffle from the inlet, and these two factors have a mutual influence on each other. The optimal conditions to maximize settling tank removal efficiency were found to be an inlet elevation of 0.87 m, a baffle distance of 0.77 m, and a baffle height of 0.56 m. According to the model, a removal efficiency of 80.6% was predicted under these conditions. In order to verify the optimal values, a three-dimensional geometry plan was created with optimal conditions and run again, resulting in a removal efficiency of 79.6%. This is close to the predicted value by the model. The results indicate that the use of a baffle can increase the efficiency of settling tanks, but careful consideration should be given to the selection of baffle height, baffle position and distance from the inlet. Improper baffle design and installation can have a negative effect on the tank removal efficiency.

## References

1. Shahrokhi, M., F. Rostami, M.A. Md. Said, S.R.S. Yazdi, and Syafalni, (2013). Experimental investigation of the influence of baffle position on the flow field, sediment concentration, and efficiency of rectangular primary sedimentation tanks. Journal of Hydraulic Engineering,. 139(1): p. 88-94.
2. Shahrokhi, M., F. Rostami, M.A.M. Said, and S.R.S. Yazdi, (2012). The effect of number of baffles on the improvement efficiency of primary sedimentation tanks. Applied Mathematical Modelling,. 36(8): p. 3725-3735.
3. Borna, M., A. Janfeshan, E. Merufinia, and A. Asnaashari, (2014). Numerical simulations of distribution and sediment transmission in pre-settled pools using Finite Volume Method and comparison with experimental results. Journal of Civil Engineering and Urbanism,. 4(3): p. 287-292.
4. Rad, M.J., P.E. Firoozabadi, and F. Rostami, (2022). Numerical Investigation of the Effect Dimensions of Rectangular Sedimentation Tanks on Its Hydraulic Efficiency Using Flow-3D Software. Acta Technica Jaurinensis,. 15(4): p. 207-220.
5. Hirom, K. and T.T. Devi, (2022). Application of computational fluid dynamics in sedimentation tank design and its recent developments: A review. Water, Air, & Soil Pollution,. 233: p. 1-26.
6. Shahrokhi, M., F. Rostami, M.A.M. Said, S.-R. Sabbagh-Yazdi, S. Syafalni, and R. Abdullah, (2012). The effect of baffle angle on primary sedimentation tank efficiency. Canadian Journal of Civil Engineering,. 39(3): p. 293-303.
7. Tarpagkou, R. and A. Pantokratoras, (2014). The influence of lamellar settler in sedimentation tanks for potable water treatment—A computational fluid dynamic study. Powder Technology,. 268: p. 139-149.
8. Ekama, G. and P. Marais, (2004). Assessing the applicability of the 1D flux theory to full-scale secondary settling tank design with a 2D hydrodynamic model. Water research,. 38(3): p. 495-506.
9. Gharagozian, A., (1998). Circular secondary clarifier investigations using a numerical model., University of California, Los Angeles.
10. Shahrokhi, M., F. Rostami, and M.A.M. Said, (2013). Numerical modeling of baffle location effects on the flow pattern of primary sedimentation tanks. Applied Mathematical Modelling,. 37(6): p. 4486-4496.

11. Razmi, A.M., R. Bakhtyar, B. Firoozabadi, and D.A. Barry, (2013). Experiments and numerical modeling of baffle configuration effects on the performance of sedimentation tanks. Canadian Journal of Civil Engineering,. 40(2): p. 140-150.
12. Liu, Y., P. Zhang, and W. Wei, (2016). Simulation of effect of a baffle on the flow patterns and hydraulic efficiency in a sedimentation tank. Desalination and Water Treatment,. 57(54): p. 25950-25959.
13. Saeedi, E., E. Behnamtalab, and S. Salehi Neyshabouri, (2020). Numerical simulation of baffle effect on the performance of sedimentation basin. Water and environment journal,. 34(2): p. 212-222.
14. Miri, J.K., B. Aminnejad, and A. Zahiri, (2023). Numerical Study of Flow Pattern, Sediment Field and Effect of the Arrangement of Guiding Blades (Baffles) on Sedimentation in Pre-Sedimentation Basins by Numerical Models. Water Resources,. 50(1): p. 68-81.
15. Heydari, M.M., M. Rahbani, and S.M.M. Jamal, (2014). Experimental and numerical investigations of baffle effect on the removal efficiency of sedimentation basin. Advances in Environmental Biology,: p. 1015-1022.
16. Guo, H., S.J. Ki, S. Oh, Y.M. Kim, S. Wang, and J.H. Kim, (2017). Numerical simulation of separation process for enhancing fine particle removal in tertiary sedimentation tank mounting adjustable baffle. Chemical engineering science,. 158: p. 21-29.