International Journal of Science and Research (IJSR)
ISSN: 2319-7064
Index Copernicus Value (2016): 79.57 | Impact Factor (2017): 7.296

Optimization of Sedimentation Tank by CFD

Sudheer Pandia[1], Dr Bharat Jhamnani[2]

[1] Master of Technology Scholar, Department of Environmental Engineering, Delhi Technological University, Shahbad Daulatpura, Main Bawana Road, Delhi, 110042, India

[2] Assistant Professor, Department of Environmental Engineering, Delhi Technological University, Shahbad Daulatpura, Main Bawana Road, Delhi, 110042, India

Abstract: ***Sedimentation tank is an important unit in water and wastewater treatment. Sedimentation tank separates the sediments from water and wastewater. The efficiency of sedimentation tank depends upon the dimensions of sedimentation tank, inlet velocity and particle diameter. It is so hard to make different types of sedimentation tank and analyses experimentally. The present study is carried out to optimize the sedimentation tank by using CFD (computational fluid dynamics) and we trace the particle trajectories by using DPM (Discrete Phase Model). The graphs of velocities, pressure drop are analyzed and compared. Efficiency of particle tracing is compared between with or without a baffle wall by varying particle diameter and the number of injected particles.***

Keywords: Sedimentation tank, Baffle wall, Computational fluid dynamics (CFD), Discrete phase model) DPM**,** Particle tracking efficiency

1. Introduction

1.1 Sedimentation tank

Sedimentation tank is very important unit in water and wastewater treatment plants. It works on the principle of gravity. In sedimentation tank if the velocity of flow is low then particles having a tendency to move towards bottom due to gravity and settle down. Therefore, a sludge layer forms in the bottom of sedimentation tank. The efficiency of sedimentation tank depends on the following points.

- Dimensions of the tank
- Inlet velocity
- Flow rate
- Particle diameter
- Particle density etc

1.2 Working principle and governing parameter

The sedimentation of discrete particles has based on the Stokes law. According to this law, the final velocity or the terminal velocity of a particle is constant in a sedimentation tank. It is achieve when the frictional force counter the force due to gravity. This terminal velocity is reach very quickly in liquid. The terminal velocity given by

$$V = \frac{d^2(\rho_\omega - \rho_o) r\omega^2}{18\eta} \quad \text{............(1)}$$

Where
d = diameter of the particle (m)

Density of wastewater does not change significantly due to temperature change. Therefore, we neglect this and standard value of 1000 kg/m3 has adopted. But the variation of viscosity depends on the temperature.

The diameter is proportional to the square root of velocity of particle so by increasing the size of particle terminal velocity increases significantly .so with a higher size of particles removal is so faster.

Our purpose is to know the benefits and application of CFD in wastewater treatment so we can improve efficiency and optimize our plants specifically in the field of sedimentation tank. (Shilton, Glynn et al. 2010). CFD is a powerful tool and its application is widely used in different kinds of industries. . In water treatment plants sedimentation is very important and common unit. The use of sedimentation is to remove the particles by the action of gravity; a major part of suspended solids is remove.

The efficiency of sedimentation tank depends on the particle settling so it is an important parameter in designing of sedimentation tank. So our purpose in this study to optimize the settling of particles.

Sedimentation is a very common unit in wastewater treatment so many research studies and papers has published on sedimentation tank. Larsen first used the applications of CFD in secondary clarifiers. Shamber and Larock[1] solved the Navier-Strokes equation, the k–e turbulence model equations by using finite volume method and done modelling of secondary clarifier by using transport equation for settling of particles.. Zhou et al.[2] Stimulate the clarifier model and considered the neutral density and hot water by interrelating energy equation with navier stroke equation.

Imam et al.[3] considered a constant settling velocity of particles in a sedimentation tank that is simply the average of settling velocities of different sizes of particles. Stamou et al.[4] Also have done work on primary sedimentation tank but he used a 2D model, he simply analysed and stimulate some equations like momentum equation and also used equation of concentrations of particles. Adams and Rodi[5] also done work on the same model but their field of work is different, they investigate the effect of inlet arrangements although it is an important parameter and instead of a normal flow, they have done their work on flow through curves. Lyn et al.[6] Investigate the flocculation process and taken six different sizes and velocities. This paper gave us some important results and very important in wastewater treatment.

Goula et al.[7] In addition, done almost same research but his work area is potable water instead of wastewater, his model deals with sedimentation tank for potable water and he investigated the effect of baffles and dependence on temperature at inlet. The drawback of his study is he did not consider the interaction between the solid phase and liquid phase, as we know that interaction affects the whole study. Wang et al.[8] done his work on rectangular sedimentation tank. He simulated the concentration of particles and flow pattern.

Sedimentation is very common and important process and it is use in industries. Kahane et al.[9] have done work in industries to reduce the operating costs. White et al.[10] Also done his work on the flow behaviour of fluid. For improving the operations in thickener, flocculation studies have done by the Farrow et al.[11]

Righetti and Romano[12] , Sbrizzai et al[13]. , Hetsroni[14] , Li et al.[15] , Reeks[16] , these researchers have investigated the interaction between solid and liquid phase but their investigation in not useful for us because their investigation is in different filed ,they were not analyse the sedimentation tank. Although on settling tank there are many researchers done their work, on settling and removal of particles. For potable water, the application of CFD models are not used much and in this field, the work on CFD is limited. The interaction between the solid and liquid phase and changes in the velocity due to that interaction and momentum change has firstly investigated by Roza Tarpagkou[17] in his research paper.

As one pioneer in numerical simulation of sedimentation tank, Larsen (1977) applied CFD simulation to several sedimentation tanks. Although with simplification and conceptualization, he still shown several major hydraulic phenomena of sedimentation tank, such as “density waterfall. Due to heavier fluid sink into bottom of sedimentation tank soon after entering, bottom current and surface return current. Nowadays, thanks to effort of computer engineers, mathematicians and fluid dynamics scientists, several more advanced models has been developed and available in commercial CFD software, based on these models and today’s high performance computer, we could run advanced simulation which far beyond than 1970’s.

Goula et al (2008) researched influence of baffle on sedimentation tank in potable water treatment by using CFD; a circulation zone is detect in the original tank without baffle. After equipped baffle, the recirculation zone around inlet in original tank decreased, the baffle enhanced setting of particles. Due to effect of baffle, particles around inlet move downwards and reach the bottom of the tank.

Density current, or turbidity current, means when two fluids with different density due to temperature, concentration or salinity confront each other, fluid with higher density sinks and flow along the bottom of fluid with smaller density, Goula et al (2008) studied the influence of temperature variation on density current in sedimentation tank. He found temperature difference between incoming fluid and fluid in tank could leads to density current. Under density current phenomenon, a rising buoyant plume appears in the tank, and changes the direction of the main circular current.

Shahrokhi et al (2012)[18] studied effect of baffles on sedimentation tanks, result show that baffle at optimum location could reduce the circulation zone, kinetic energy and maximum velocity magnitude; uniform velocity vector inside the settling zone from CFD simulation result could indicate better sedimentation effect.

2. Problem Description

2.1 Geometry of sedimentation tank

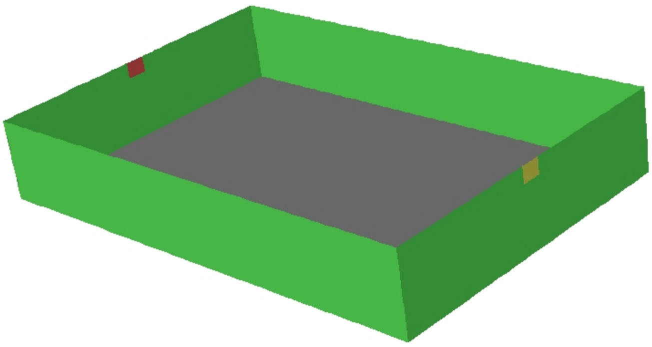

Figure 1: Geometry of sedimentation tank without baffle wall

Table 1: Geometry details

Depth	Length	Width	Inlet Channel	Outlet Channel
.8	6	4	.25*.2	.25*.2

In the above figure inlet has shown in yellow colour, outlet has shown in red colour, walls are in green colour, and bottom wall is in grey colour. The noticeable point is, it seems an open tank but in actual case, it is covered. The purpose of showing it uncovered is to show its inner features that can be see easily in the above figure.

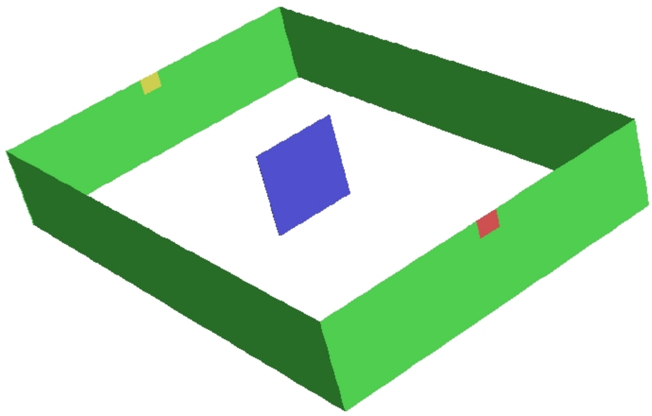

Figure 2: Geometry of sedimentation tank with baffle wall

All the dimensions are similar as shown in sedimentation tank without baffle instead of a baffle provided on mid plane across the length shows in figure by blue colour.

2.2 Grid generation and dependence study

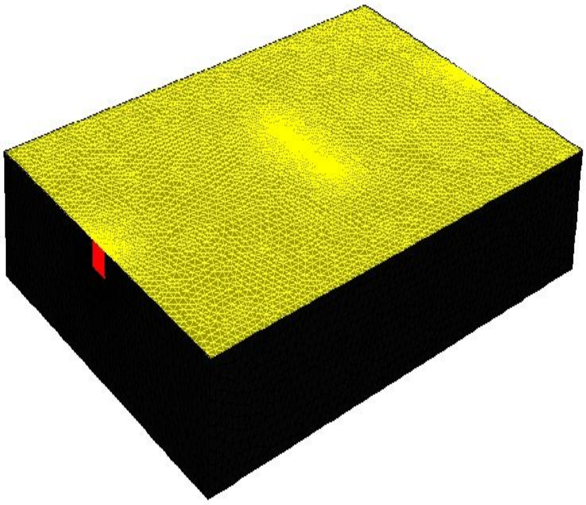

Figure 3: Meshing of sedimentation tank without baffle wall

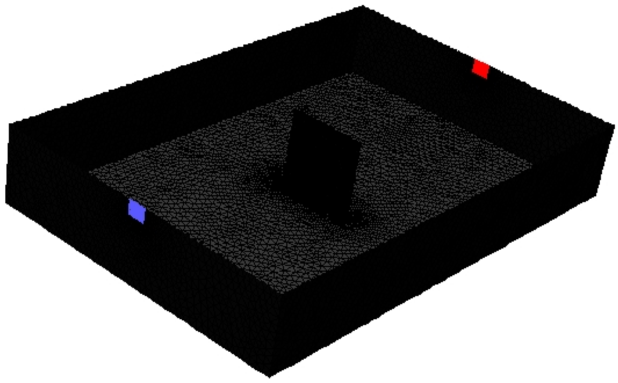

Figure 4: Meshing of sedimentation tank with a baffle wall

Table 2

Meshing details	cells	nodes	faces
Geometry without baffle	554836	199592	1272520
Geometry with baffle	1921912	337036	2122784

Mesh independent study

The aim of this study is to show that the results of the CFD simulation will not change with our meshing if we have done the meshing perfectly. In our actual, meshing or selected meshing there are 554836 cells but in our finer mesh, there are 322546 cells only. So as shown in the below graph the results will not change with different meshing so our study is mesh independent. It is noticeable that the graph is plot between average velocity on mid plane Vs the distances along x- axis or the length of the sedimentation tank.

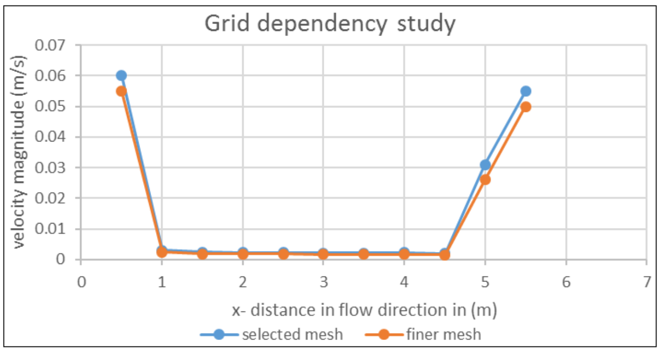

Figure 5: Graph of grid dependency

3. Methodology

Mathematical Model

By using Euler–Euler or an Euler–Lagrange approach we can analyse the hydrodynamic behaviour of a sedimentation tank, from the above approaches we can easily analyse the multiphase behaviour in a sedimentation tank. Eulerian applications are widely used in the analysis of sedimentation tank but the major drawback of this approach is it does not consider the individual particle motion as in lagrangian approach. Due to wide range of applications, lagrangian approach is majorly use in multiphase problems. In Lagrangian approach every individual particle is consider so it gives a realistic and a well-defined model by the use of this approach, we can simulate the flow behaviour in multiphase and a realistic way. In multiphase approach particle or solid phase is treated in the lagrangian way and fluid is always treated as continuum phase. In this phase(multiphase) coupling effect may be considered ,due to coupling effect the model will become more difficult but the results are more realistic.in these types of models we assume that the volume of solid phase is much less than the volume of primary phase or liquid phase although high loading rate is allowed. According to De Clercq et al., we cannot apply the lagrangian approach if the volume of secondary phase is high that means that we can apply that approach if volume of secondary phase does not exceeds 10–12%. As mentioned earlier in this approach the particle motion and their paths are individually consider at different intervals of time.

RNG k-ε model

Equation of kinetic energy

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial(\rho k u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_k}\frac{\partial k}{\partial x_j}\right]+2\mu_t E_{ij}E_{ij}-\rho\varepsilon \quad (2)$$

Equation of dissipation

$$\frac{\partial(\rho\varepsilon)}{\partial t}+\frac{\partial(\rho\varepsilon u_i)}{\partial x_i}=\frac{\partial}{\partial x_j}\left[\frac{\mu_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}2\mu_t E_{ij}E_{ij}-C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}$$

………………..(3)

Governing equations

By the application of integration on the equation of force, balance of discrete phase trajectory our model works that is lagrangian approach. In this equation, we balance all the forces acting on the particle like inertia force.

$m_p\,(du_p/dt) = m_p F_D (u-u_p) + m_f (Du/Dt) + 1/2\, m_f (Du/Dt - du_p/dt) + (m_p - m_f)g + 1/2\,(\pi\, \rho_p r^2)$ ……………….(4)

C_L LV2

Where:

$F_D = 18\mu\, C_D\, Re/ (24\, \rho_p\, d_p^{2})$ ……………………….(5)

d_p is diameter of particle

Fluid phase: By the application of navier stroke equation, fluid phase is consider as continuum. The following equation accounts the non-conservation of mass and non-conservation of energy and incompressibility also.

$\partial u_i / \partial x_i = 0$

$$U_j \partial U_i/\partial x_j = -1/\rho\, \partial p/\partial x_i + \partial/\partial x_i \left(v(\partial U_i/\partial x_j + \partial U_j/\partial x_i) - u_i' u_j'\right) \quad ……(6)$$

Turbulence. The RNG k–e turbulence model is show in the following equation

$$\partial/\partial X_i\, (\rho k u_i) = \partial/\partial X_j (a_k \mu_{eff}\, \partial k/\partial X_j) + G_k + G_b - \rho\, \varepsilon \quad ……(7)$$

$$\partial/\partial X_i\, (\rho \varepsilon u_i) = \partial/\partial X_j (a_\varepsilon \mu_{eff}\, \partial \varepsilon/\partial X_j) + C_{1\varepsilon}\varepsilon/K(G_k + C_{3\varepsilon} G_b) - C_{2\varepsilon}\, \rho\, \varepsilon^2/K - R_\varepsilon \quad ………………………..(8)$$

Momentum exchange: As mentioned above in lagrangian approach we use multiphase or discrete phase with or without coupled effects but we are considering coupling effect also. momentum exchange in multiphase is between the primary and secondary phase that are liquid and solid phase .we will consider the momentum transfer from primary to secondary and vice versa . In single-phase simulations, we does not consider the momentum exchange between the two phases. Multiphase simulations gives better and realistic results than the single phase

This momentum change is compute as:

$$F = \sum \left(18\mu C Re(u_p - u)/(\rho_p d_b^2 24) + f_{other}\right) m p \Delta t \quad ….(9)$$

4. Results and Discussion

4.1 Model validation

For validate the model we make a mid-plan along the length of the sedimentation tank and by simulation we calculate the velocity magnitude at different locations like x= 1, 2, 3, 4, 5, 6 and then draw the results on the graph and compare that results with numerical of Kantoush et al (CCHE2D). We can see in the following graph that results are matching.

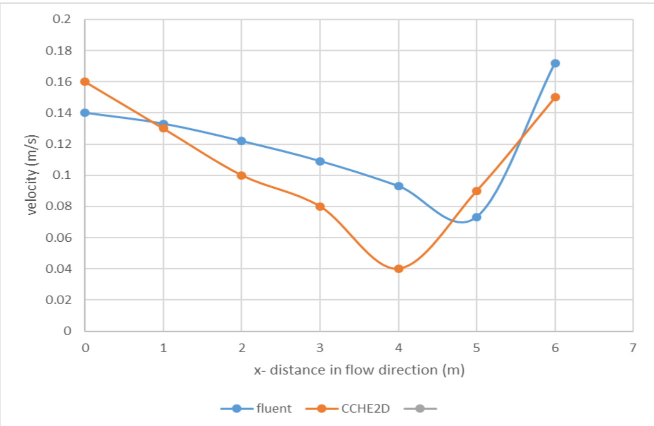

Figure 6

4.2 Comparison between velocity contours

This study can easily compare the simulations of velocity contours of sedimentation tanks with or without baffle wall. As shown in below figures the colourful scale in the left of the figure shows the velocity magnitude and by the help of this scale, we can easily see the variation of velocity in the two profiles. It is noticeable that we are considering the middle plane along the length so all the results that are show below are on the middle plane.

It is easily shown in the below figures that in the tank with baffle ,after the baffle the profile is changed and velocity is decreased that's why more sedimentation takes place in case of a sedimentation tank with baffles

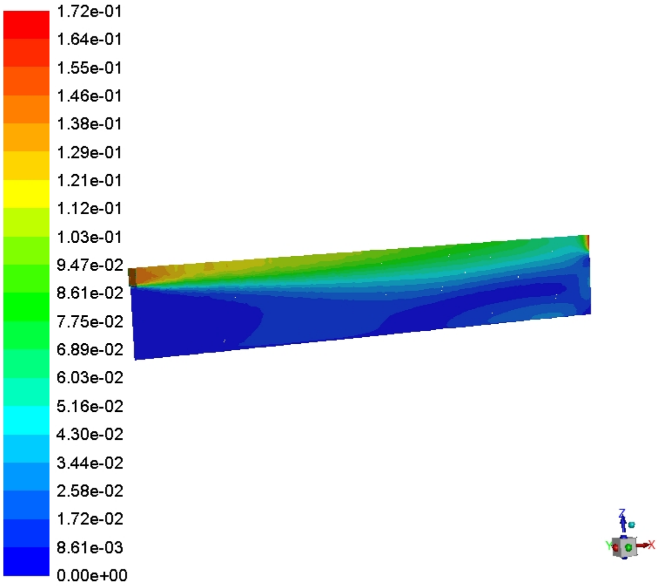

Figure 7: Velocity contour without a baffle wall

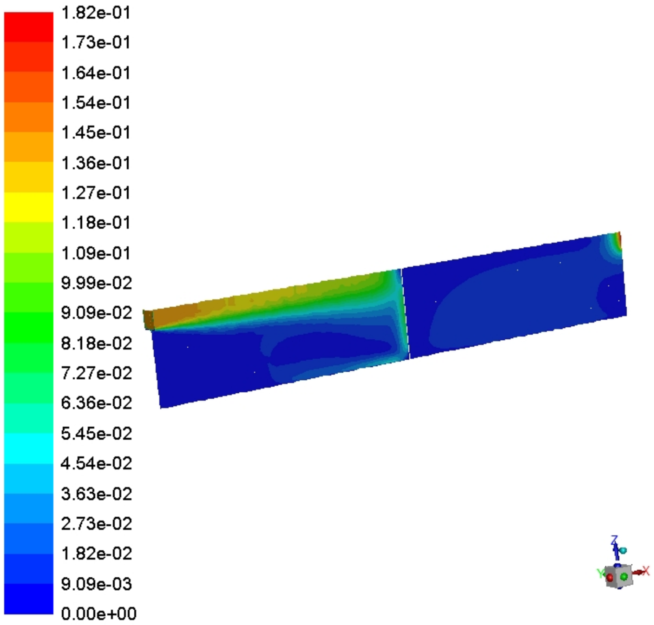

Figure 8 Velocity contour with a baffle wall

4.3 Comparison between pressure contours

Although there is a wide application of these pressure contours in designing of the walls of a sedimentation tank. However, we focus on the comparative study or the change in pressure contour by the application of a baffle. At the location of baffle, pressure is higher than the other locations. It is due to the turbulence created at the baffle.

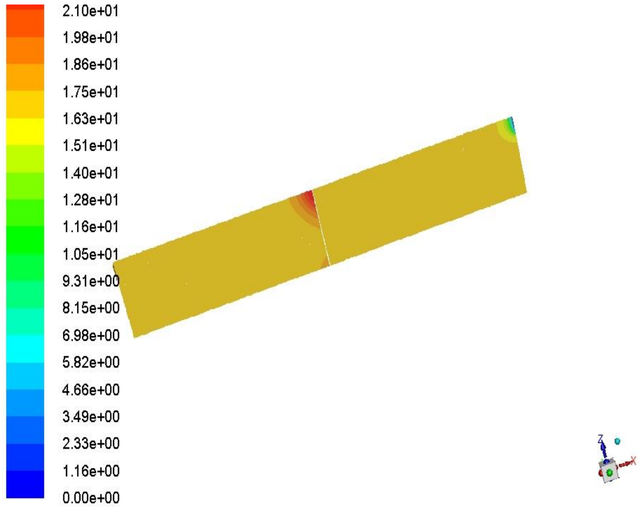

Figure 9: Pressure contour of sedimentation tank with baffle

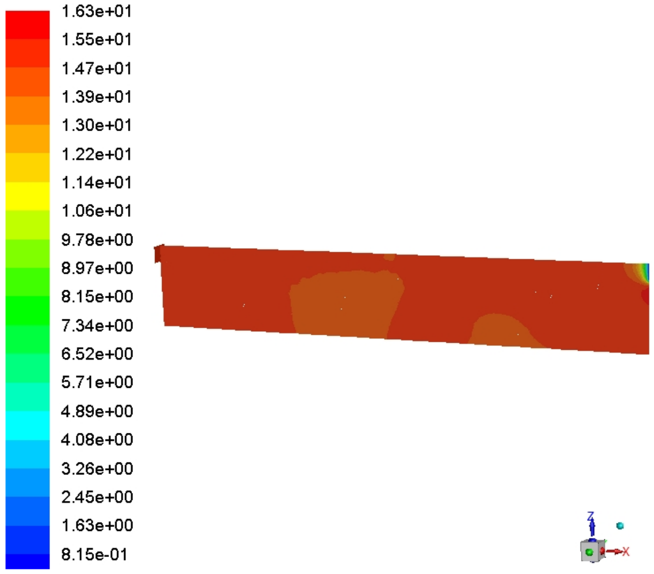

Figure 10: Pressure contour of sedimentation tank without baffle

4.4 Path lines

Path lines are the trajectories of flow of fluid if we can consider individual elements in the fluid. As shown in the below figure it is clearly shown that in the case of baffle wall higher turbulence will generate due to loss of energy occurs and velocity diminishes by a considerable amount so less velocity leads to higher sedimentation and higher efficiency. Without baffle fluid moves in a defined manner but in case of baffle, the randomness will increase and path lines become congested that means a particle affect the motion of others.

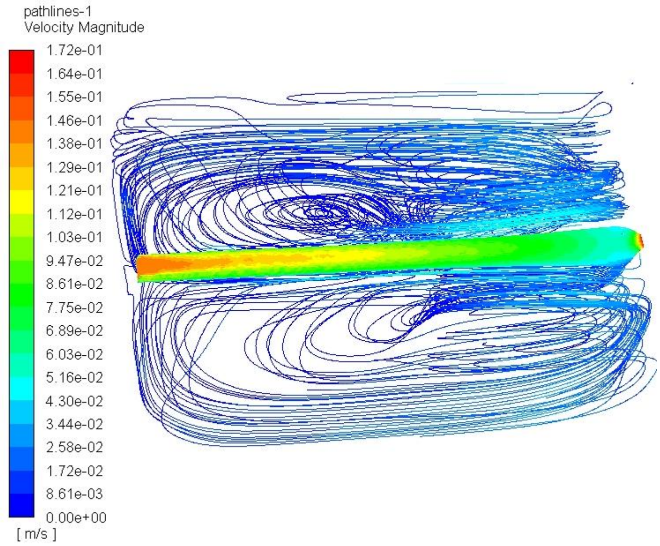

Figure 11: Path lines without baffle wall

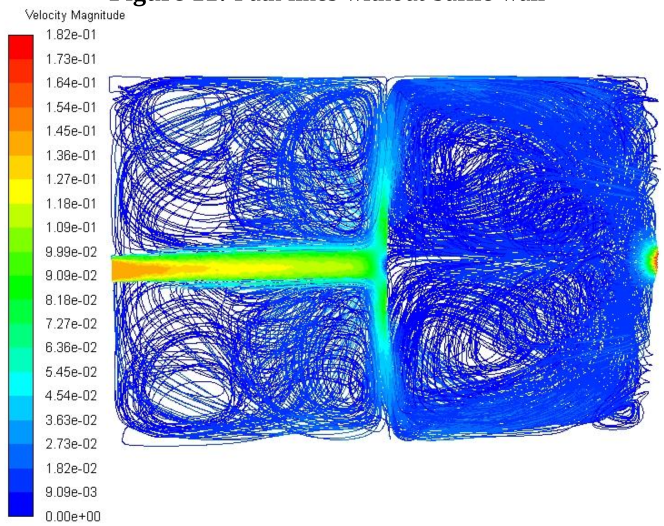

Figure 12: Path lines with a baffle wall

4.5 Particle tracking

Our main objective of sedimentation tank is to remove maximum suspended particles or sediments. In the below figures it is clearly understood that by the use of a baffle wall we can increase the efficiency of a sedimentation tank. By the use of a baffle, velocity will decrease significantly and residence time of particles will increase consequently more sedimentation takes place. The results of simulations of particle tracking are such that if we inject 246 particles from inlet in a sedimentation tank. Without baffle wall only 91 particles are trapped and the remaining are escaped but in case of sedimentation tank with baffle wall 216 particles are trapped from 246 and remaining are escaped therefore by the use of a baffle we can increase the efficiency of the particles tracking . We can also check the efficiency of the sedimentation tank with varying the velocity inlet, particle diameter, or by the changing, the dimensions of the sedimentation tank. Therefore, by the use of CFD analysis we can optimize the sedimentation tank. The efficiency is 37% of the sedimentation tank without a baffle wall and with baffle wall, efficiency will increase to 85.37%.

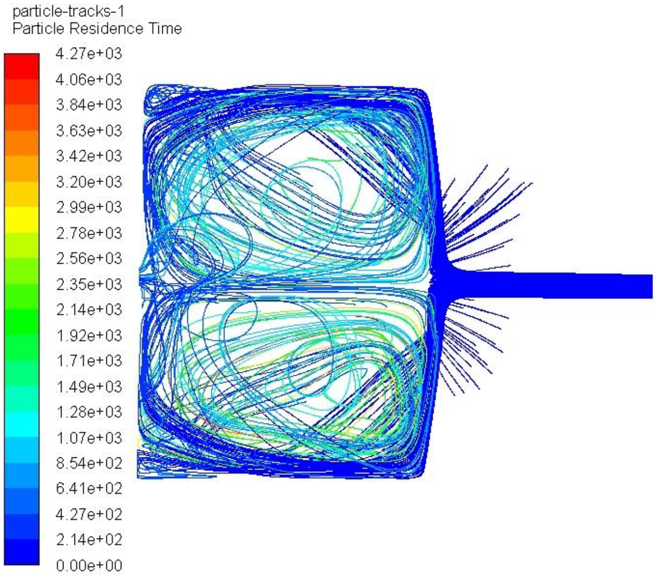

Figure 13: Particle tracking with baffle wall

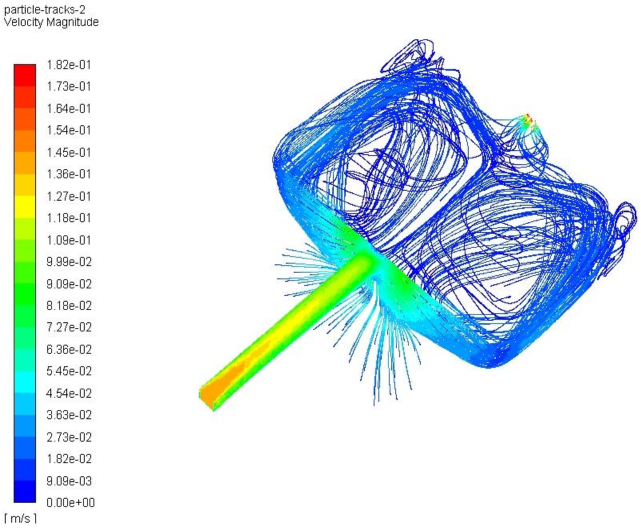

Figure 14: Particle tracking with baffle wall and velocity magnitude

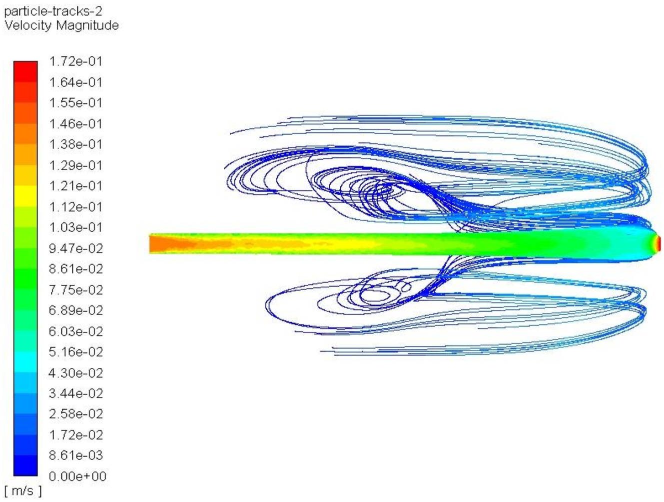

Figure 15: Particle tracking with velocity magnitude without a baffle wall

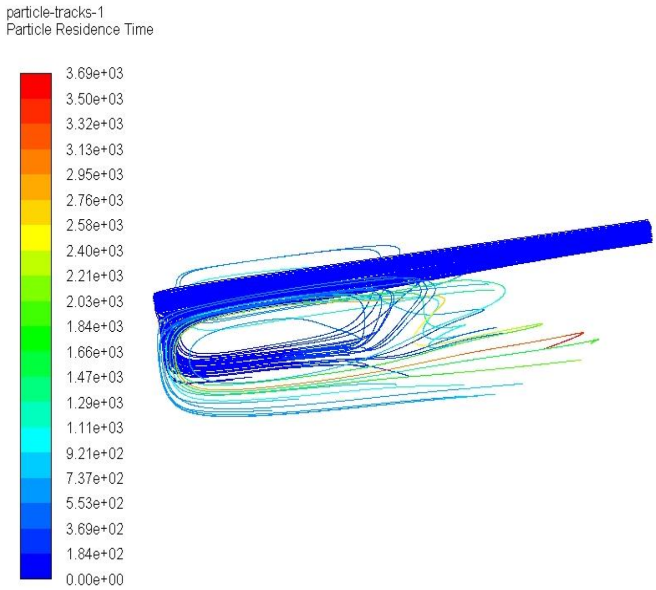

Figure 16: Particle tracking with residence time without a baffle wall

4.6 Streamlines

Streamline show the direction of flow of fluid. In case of sedimentation tank with baffle wall the flow of fluid has interrupted. Due to this disturbance, turbulence will generate subsequently velocity of fluid decrease. As we know sedimentation is inversely proportional to the velocity, therefore, particle tracking will rise and efficiency of tank would increase.

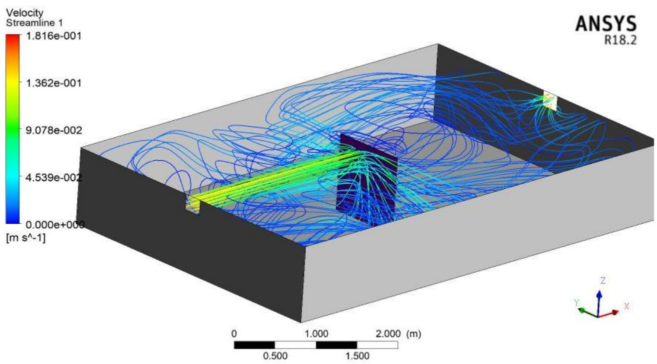

Figure 17: Streamlines of a sedimentation tank with a baffle wall

4.7 Particle tracking with varying diameter

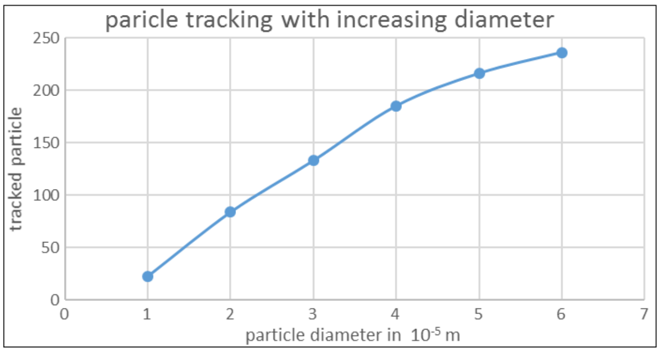

Figure 18

As shown in the graph with the increase in particle diameter sedimentation will increase. In the above graph diameter of particles has shown on x-axis and their dimensions are in 10^{-5} m. sedimentation or particle trapping will increase with size due to the effect of gravity. In addition, no. of particles has shown on y-axis.

4.8 Efficiency of tank with varying particle diameter

This graph shows the efficiency with increasing the diameter of the particle. The total injected particles are 246 at inlet . It is very clear that efficiency will increase with diameter and it will go up to 100% if we further increase the diameter of the particle.

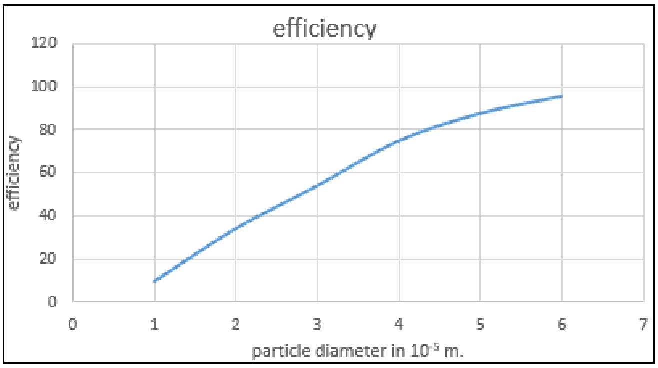

Figure 19: Graph between particle injected and trapped

This graph shows the trapped particles if injected particles are varying. In the below graph two graphs are showing, in which the graph of baffle shows more trapping. However, the graph is not linear and in case of baffle wall the trapped particles will increase with the injected particles. It can be note down by the slopes between two points.in case of the tank without baffle the trapping will almost linear because in the case of baffle turbulence will create and a particle will affect the movement of other particle so particle trapping will increase.

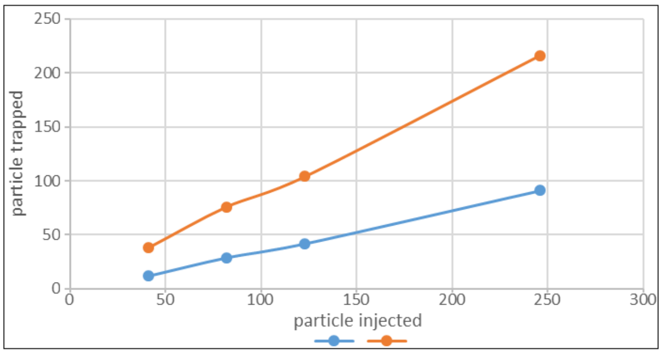

Figure 20

5. Conclusions

In order to attain the aims stated above, a good knowledge of sedimentation tank and CFD required. CFD can easily applied in the optimization of sedimentation tank. Particle tracking in sedimentation tank can be achieve for different inlet velocities and for different particle diameters. CFD is an easy and inexpensive tool for optimizing a sedimentation tank. By the use of CFD, we can easily simulate the performance of sedimentation tank.

Baffle can be installed easily and economically without significant influence on the operation. Single phase CFD simulation result shows that baffle re-distribute incoming wastewater on the whole cross-section, reduced surface current and re-circulation regions. By displaying contour of velocity magnitude, and velocity vectors, we found baffle in suitable location and tilted bottom reduces the circulation zone and kinetic energy, create uniform velocity vector inside the sedimentation tank. Therefore, the baffle and tilted bottom improved the hydraulic efficiency of the original sedimentation tank.

In this study, we find out the contours of velocity and pressure on the mid plane, streamlines and particles tracking with either velocity magnitude or residence time. We simulate the sedimentation tank with a baffle wall also. If we compare the efficiency of two sedimentation tank then efficiency is 37% of the sedimentation tank without a baffle wall and with a baffle, wall efficiency will increase to 85.37%. This increment in efficiency is due to the reduction of velocity and more residence time, which leads to more sedimentation.

All in all, no matter in physical, chemical and biological treatment processes, as well as other components in wastewater treatment, such as disinfection, sludge treatment, odour control, pumping station or even sewer line optimization, CFD can be used in all of these processes. CFD will bring new concepts and ideas for water and wastewater industries, change the traditional design methodology of water and wastewater devices, in the foreseeable future, we will see more and more applications of CFD in water and wastewater treatment field.

References

[1] D.R. Shamber, B.E. Larock, Numerical analysis of flow in sedimentation basins, J. Hydr. Div, ASCE. 107 (1981) 575–591.
[2] S. Zhou, J.A. McCorquodale, A.M. Godo, Short-circuiting and density interface in primary clarifiers, J. Hydr. Eng. 120 (1994) 1060–1080.
[3] E. Imam, J.A. McCorquodale, J.K. Bewtra, Numerical modelling of sedimentation tanks, J. Hydr. Eng. 109 (1983) 1740–1754.
[4] A.I. Stamou, E.A. Adams, W. Rodi, Numerical modelling of flow and settling in primary rectangular clarifiers, J. Hydr. Res. 27 (1989) 665–682.
[5] E.W. Adams, W. Rodi, Modelling flow and mixing in sedimentation tanks, J. Hydr. Eng. 116 (1990) 895–913.
[6] D.A. Lyn, A. Stamou, W. Rodi, Density currents and shear induced flocculation in sedimentation tanks, J. Hydr. Eng. 118 (1992) 849–867.
[7] A.M. Goula, M. Kostoglou, T.D. Karapantsios, A.I. Zouboulis, 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 (2008) 110–121.

[8] X. Wang, L. Yang, Y. Sun, L. Song, M. Zhang, Y. Cao, Three-dimensional simulation on the water flow field and suspended solids concentration in the rectangular sedimentation tank, ASCE 134 (2008) 902–911
[9] R. Kahane, T.V. Nguyen, M.P. Schwarz, CFD modelling of thickeners at Worsley Alumina Pty Ltd, Appl. Math. Model. 26 (2002) 281–296.
[10] R. White, T.V. Nguyen, I. Sutalo, Modelling studies of fluid flow in a thickener feedwell model, Miner. Eng. 16 (2003) 145–150.
[11] J.B. Farrow, P.D. Fawell, R.R.M. Johnston, T.B. Nguyen, M. Rudman, K. Simic, J.D. Swift, Recent developments in techniques and methodologies for improving thickener performance, Chem. Eng. J. 80 (2000) 149–155.
[12] M. Righetti, G.P. Romano, Particle–fluid interactions in a plane near-wall turbulent flow, J. Fluid Mech. 505 (2004) 93–121.
[13] F. Sbrizzai, V. Lavezzo, R. Verzicco, M. Campolo, A. Soldati, Direct numerical simulation of turbulent particle dispersion in an unbaffled stirred-tank reactor, Chem. Eng. Sci. 61 (2006) 2843–2851
[14] G. Hetsroni, Particles-turbulence interaction, Int. J. Multiphase Flow 15 (1989) 735–746.
[15] C. Li, A. Mosyak, G. Hetsroni, Direct numerical simulation of particle-turbulence interaction, J. Multiphase Flow 25 (1999) 187–200.
[16] M.W. Reeks, The transport of discrete particles in inhomogeneous turbulence, J. Aerosol. Sci. 14 (1983) 729–739.
[17] R. Tarpagkou, A. Pantokratoras / Applied Mathematical Modelling 37 (2013) 3478–3494
[18] Shahrokhi, M., Rostami, F., & Said, M. A. M. (2012). Numerical modeling of baffle location effects on the flow pattern of primary sedimentation

Author Profile

Sudheer Pandia, Master of Technology Scholar, Department of Environmental Engineering, Delhi Technological University

Dr Bharat Jhamnani, Assistant Professor, Department of Environmental Engineering, Delhi Technological University