Optimization of the sedimentation tank with CFD simulation

O. Shelestina*, **H. Ratnaweera**

Norwegian University of Life Sciences, PO Box 5003-IMT, 1432 Aas, Norway
* Corresponding author. E-mail: oleksafu@nmbu.no; tel.: +47 451 90 625

Abstract

The aim of this research project was to investigate the possibilities of CFD and introduce CFD as a new tool in optimization of water treatment process, with focus on sedimentation tank. Paper consists introduction, where introduced importance of cleaned water and existed water treatment mechanisms. Sedimentation process as the simplest and effective particle separation process. Describing of methods that were used during researching. Theoretical background, governing equations and simulation procedure of CFD, turbulence and multiphase model is introduced in “Methods” chapter. In the results section, ways of modernization and new retrofitted model are represented.

Key words: Computational Fluid Dynamics (CFD), sedimentation tank, simulation, turbulence model, modernization, wastewater treatment.

Introduction

Seventy percent of the Earth’s surface is covered with water, and yet there is an acute water shortage. This is because 97.5 per cent of it is salt water, and almost three quarters of the freshwater is frozen in ice caps. Consequently, more than one third of the world’s population, that is 2.4 billion people, has no access to clean water. The issue of water - its quality and its quantity. To meet these goals water has to be treated properly.

Water treatment is the complicated and inseverable process which consists several stages depending on each other: mechanical treatment (screening, filtration), chemical treatment, sedimentation process and disinfection. Sedimentation is the most common particle separation process in the wastewater treatment. Nowadays, sedimentation process commonly used to remove impurities that have been formed during coagulation and flocculation process, flocculated particles settle to the bottom of the tank, while cleaned flow goes out of the tank, this procedure able to remove 50 -70% of the suspended solids, 25-40 % of BOD from the wastewater (Rajvaidya and Markandey,1998). Despite the practical importance of the sedimentation tanks, current design practice mostly dependence on empirical formulas which do not take full account of the detailed hydrodynamics of the system. The list if factors impact on the capacity and productivity of a sedimentation tank, such as surface and solid rates, design of the tank, what kind of solids removal mechanism, inlet outlet parameters, and etc. To account it, proper understanding of sedimentation tank behavior, processes which going inside is crucially important for effective tank design and operation. Traditional sedimentation tank design methods work like a “black-box”, we can observe only external features, but we have no possibility to look inside of the tank (Metcalf, 2002). To avoid “black-box” effect, tedious, time consuming, and expensive experiments Computational Fluid Dynamics (CFD) started used during the last decades. CFD simulates hydraulic performance of a sedimentation tank before design work starts, thus avoid probability of failures (Anderson, 1995).

The aim of this study is to show the use, benefits, applicability and main possibilities of CFD in water treatment technology with focus on sedimentation tank. As (Shilton, Glynn et al. 2010) writes: “We need only to look at other industries to see how CFD can be used reliably for complex design. It is powerful tool, whose potential has been practically untapped by the water industry. As awareness of this grows there can be little doubt that we shall be seeing much more of CFD in the future. The aim also is to validate and calibrate the model of the existing sedimentation tank which situated at the Drøbak WWTP, and to propose the retrofitted models to increase efficiency of the tank.

2. Material and Methods

2.1 Basic of CFD theoretical explanation

Equations are used to mathematically describe the physics of fluid flow. The continuity equation and the momentum equation, also known as the Navier-Stokes equation, and energy equation are needed to describe the state of any type of flow and are generally solved for all flows in CFD modelling, see equation 1 and 2, respectively (ANSYS CFX Solver Theory Guide 2011).

Continuity equation: $$\frac{\partial \rho}{\partial t} + \nabla * (\rho * u) = 0 \quad (1)$$

Navier-Stokes equation: $$\frac{\partial \rho u}{\partial t} + \nabla * (\rho u u) = -\nabla p + \nabla * \tau + \rho g \quad (2)$$

Energy Equation: $$\rho c_{\mu} \frac{\partial T}{\partial t} + \rho c_{\mu} U_i \frac{\partial T}{\partial x_i} = -P \frac{\partial U_i}{\partial x_i} + \lambda \frac{\partial^2 T}{\partial x_i^2} - \tau_{ij} \frac{\partial U_j}{\partial x_i} \quad (3)$$

Where: p – is a density, u – is an instantaneous velocity, p – is a pressure, τ- viscous strees tensor, and g – is the gravity vector.

Turbulence model

To simulate sedimentation process in this study we use standard k- ε model, also known as turbulence model. When fluid velocity increase, the Reynolds number will exceed the critical Reynolds number, so all flow will mixing and will has chaotic and unsteady properties – such flow, we call turbulent flow.

The standard k-ε model include two major equations. These are turbulent kinetic energy equation k, and equation of dissipation energy ε.

Turbulence kinetic energy: $$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + P_b - \rho\varepsilon - Y_M + S_k$$

Dissipation energy: $$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(P_k + C_{3\varepsilon}P_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} + S_\varepsilon$$

Where: μ – turbulent viscosity, P_k- production of k, P_b- effect of buoyancy, $C_{1\varepsilon}$, $C_{2\varepsilon}$, and $C_{3\varepsilon}$ – are constant, σ_k and σ_ε – are Prandtl numbers.

Multi - phase model

Multiphase flow occurs when more than one material is present in a flow field and the materials are present in different physical states of matter or are present in the same physical state of matter but with distinct chemical properties. In multiphase models, the main difficulties are due to the interface between phases, we use Mixture model as a multi-phase model during the simulation. By follow the notations of Ishii (1975) we can formulate continuity and momentum equations for each phase.

Continuity equation: $$\frac{\partial}{\partial T}(\rho_m) + \nabla(\rho_m \vec{V_m}) = m \quad (4)$$

Momentum equation: $\frac{\partial}{\partial t}(\rho_m \vec{V_m}) + \nabla(\rho_m \vec{V_m} \vec{V_m}) = -\nabla p + \nabla[\mu_m(\nabla \vec{V_m} + \nabla \vec{V_m}^{t})] + \rho_m \vec{g} + \vec{F} + \nabla * (\sum_{k=1}^{n} \propto_k \rho_k \vec{V}_{dr,k} \vec{V}_{dr,k})$ (5)

Where: $\vec{V_m}$ - is mass averaged velocity, ρ_m – is density of the mixture, μ_m- is viscosity of mixture, $\vec{V}_{dr,k}$ - drift velocity of the secondary phase.

2.2 Boundary and initial conditions of the sedimentation tank simulation

A full – scale horizontal sedimentation tank was investigated in the Drøbak wastewater treatment plant. The plant receives raw water from the houses, with capacity around 400 m^3/h. The pre-treatment process includes coagulation and flocculation, and after it water goes directly to the sedimentation tank throw two inlet pipes, with dimensional 500m each, and situated on the 2 m below the water level. There are two baffles front of the inlet on the distance of 500 m from, with dimensions 500 mm each. The sludge goes down to the bottom, and treated, water goes out of the tank throw outlet pipes. There are three outlet launders, two with 20 m length, and central one with 14,2 m, depth equal approximately 22 cm, and with is 0.5 m.

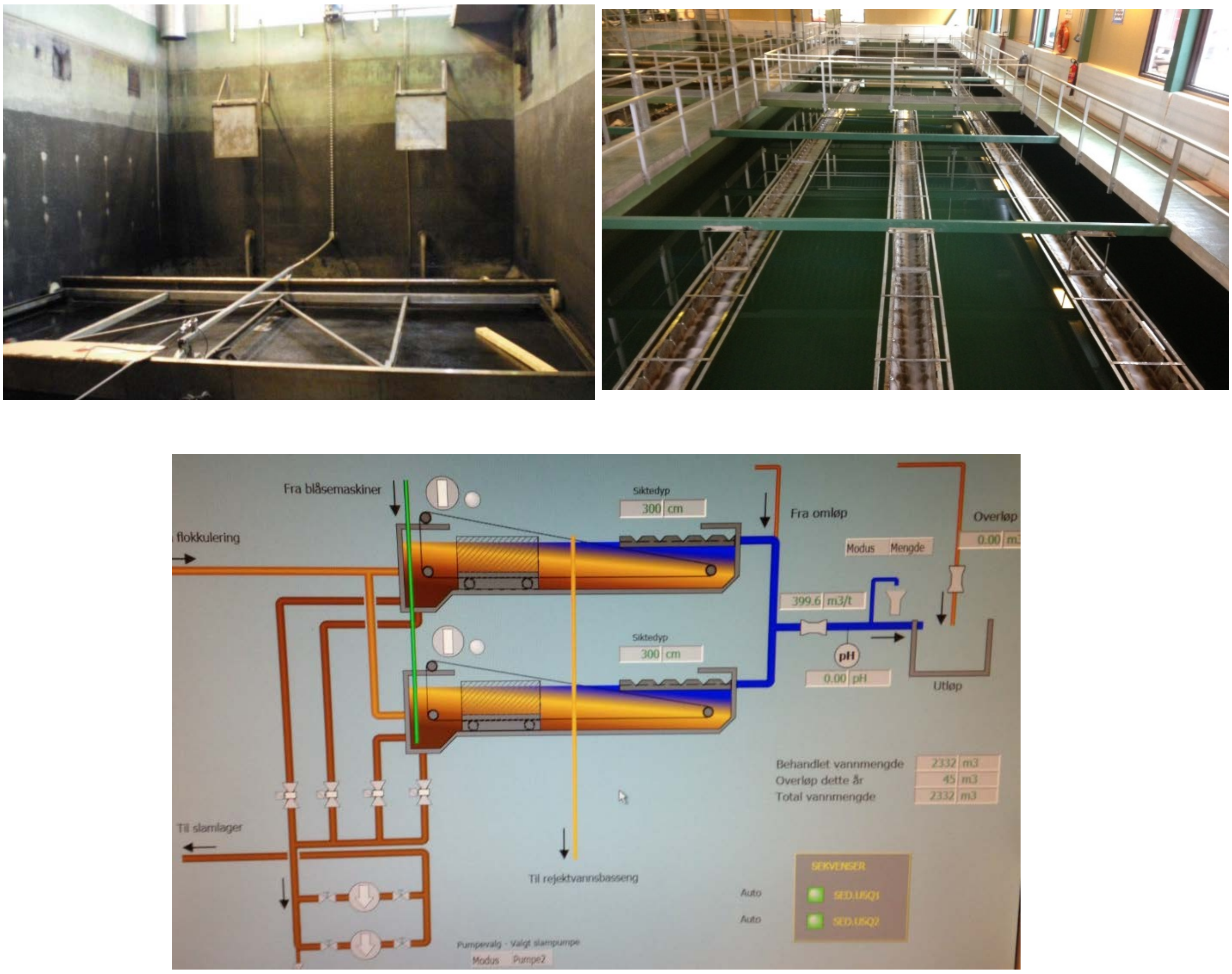


Figure 1 Sedimentation tank at the Drøbak wastewater treatment plant

To simulate the sedimentation process, we use such initial conditions as: inlet – velocity inlet, outlet – outflow. Design of the tank was created by GAMBIT software, based on the original geometry of the sedimentation tank. CFD models are used to describe the behavior of multiphase flow by Euler-Euler approach and multiphase model. The primary phase –sewage has density is 1000 kg/m^3, viscosity is 0.001 kg/m*s; Secondary phase – sediments has density 1200 kg/m^3, viscosity is 0.01 kg/m*s and diameter of particles is 0.1 mm, and volume fractions of sediments 5 %.

3. Results and Conclusions

3.1 RTD simulation for the single-phase model with the help of the Tracer test

Commercial CFD software Fluent is used in this study, in order to study the flow pattern and evaluate the hydraulic performance of the sedimentation tank. The whole procedure of the RTD simulation with the help of tracer test describes in the appendix 3. There are two types of tracer test conducting, pulse method (E-curve shape) and step method (F-curve shape) (USEPA.2003)

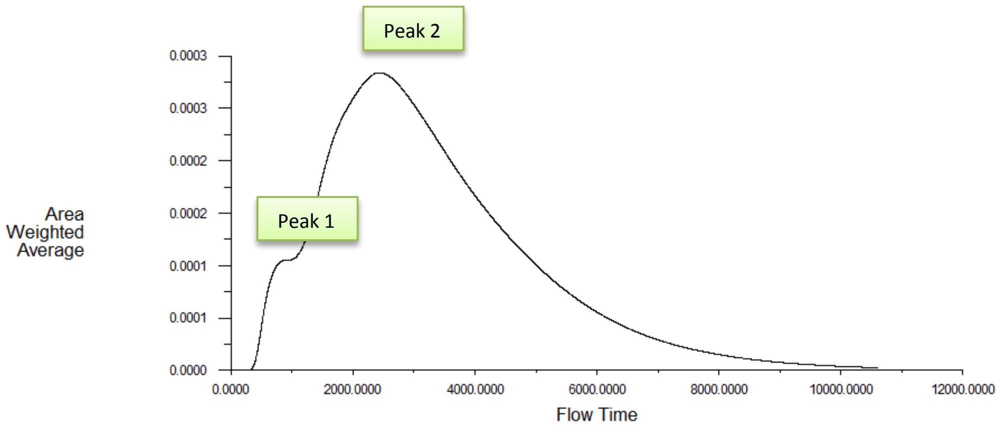


Figure 2 RTD curve of the original tank, E -type

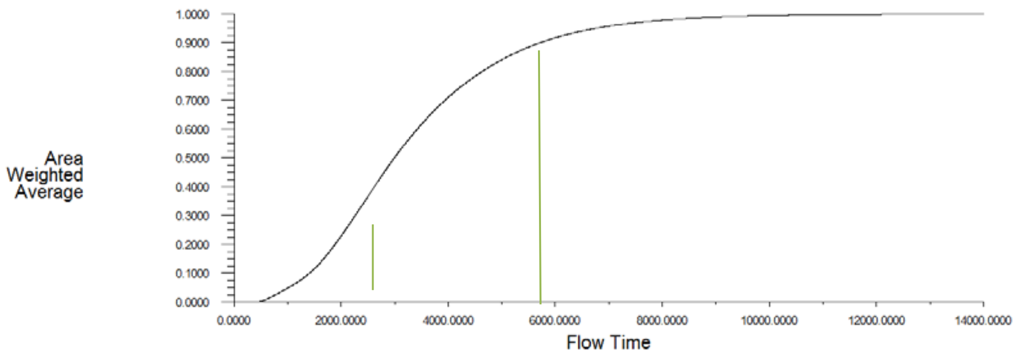


Figure 3 RTD curve of the original tank, F -type

Analysis of this two type curves shows that the RTD simulation of the original tank design coincides, and both have two peaks. In the first curve, we can observe first small, sharpest peak, which corresponds to the smallest volume due to short – circuit in the tank, and strong surface current, and we can make conclusion that t_{10} is high, is the time for 10 percent of the inlet concentration observed at the outlet. As seen from the Figure 2, the first peak range from time 0 until around 500s after tracer injected. Second peak occur approximately at the 1500s, and can be explained by re-circulating current time. Later, the RTD curve shows decreasing in a quasi – exponential fashion by the influence of strong bottom density current.

According to the figure 3 over 15 percent of tracer pass through the outlet during the first period, what point out on the short detention time.

3.2 RTD simulation based on the Drøbak WWTP

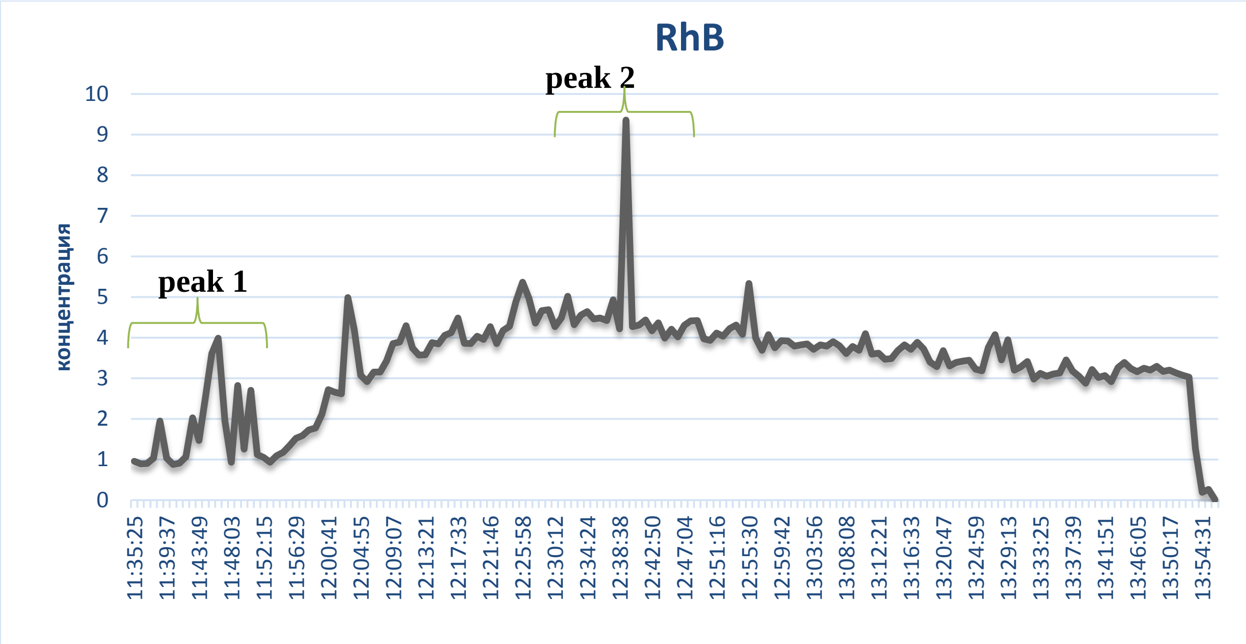


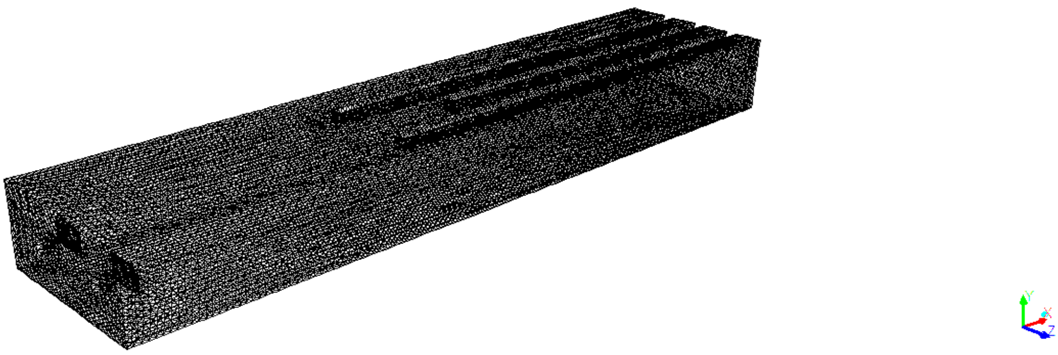
Figure 4 RTD curve based on the real Drøbak sedimentation tank

After comparison of these to curves (computational and real tracer curves), we can surely say, they have the same shape, and both of them have two peaks, which indicates that computational simulation based on the CFD software can be used for the further investigation as real results. To take a look on these curves, we can observe two peaks phenomena, what is the reason? It could be because of short –circuit in the tank, short retention time and bad tank design.

3.3 Multi-phase simulation result

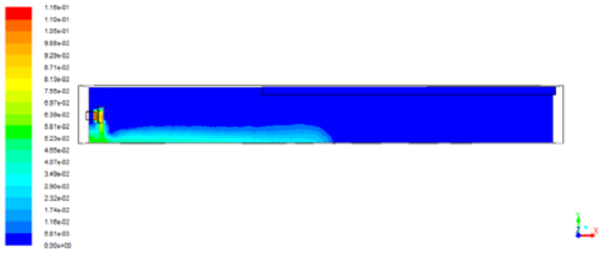
3.3.1 Multi –phase simulation result of the original sedimentation tank

Figure 5 (a) represent's geometry model of the existing sedimentation tank for the CFD simulation with the help of the GAMBIT software. All dimensionals are saved and corresponds to the Drøbak sedimentation tank. 4.

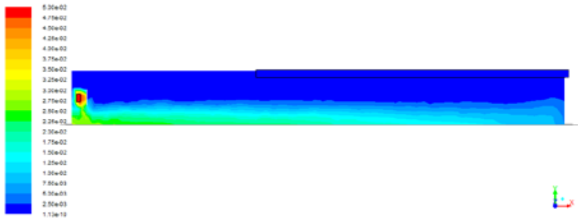


a

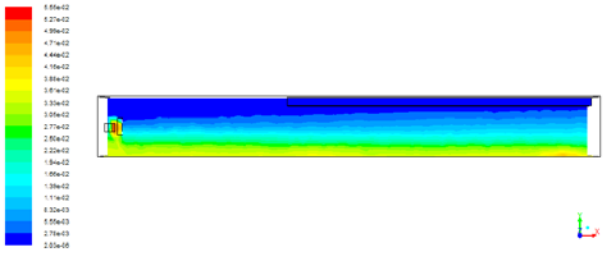
Figures (b) - (g) represent the evolution of sediments volume fractions after 100 s, 200s, 500s, 1000s, 2000s on the x-y plane at the tank bottom. The line across the bottom was used for the volume fraction plot. Distribution of the sediments volume fraction along the bottom represents on the plot (i) from the inlet (starts position -16m) to the outlet (ending position +16) at the 2000s.



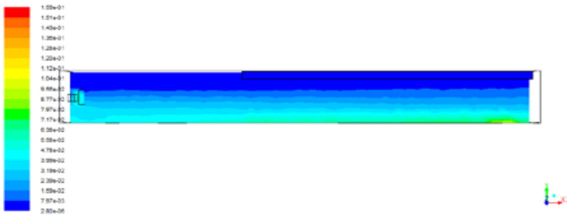
b



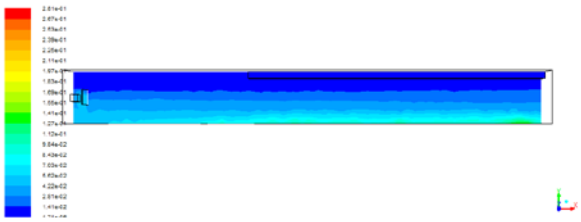
c



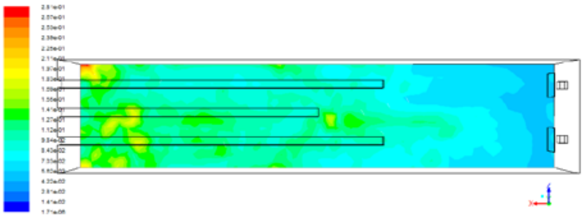
d



e

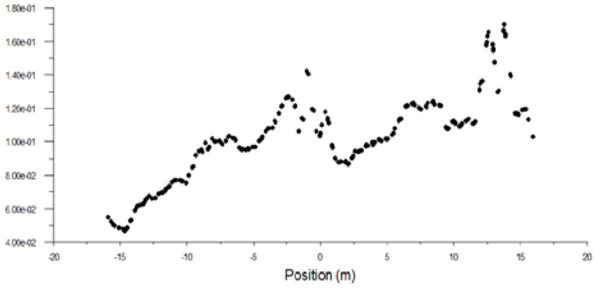


f

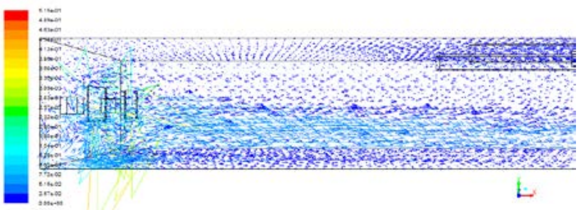


g

Figures 5 f – g show that all part of particles settle along the bottom, but biggest part of the sediments is moved to the outlet by entering flow, and part of particles can get to the treated water, what can be seen on the volume fraction plot (i). This phenomenon can be due to the small baffles, which sprays a stream on the walls, and flow velocity is not reduced, moreover figure 5.h shows return downward current, thus generating an eddy and circulation.



I



h

Figure 5 Flow distribution of the existing sedimentation tank. (b) – (g) counters of the volume distribution on the y-z plane; (i) volume fraction plot; (h) – counter of the velocity.

3.3.2 Multi –phase simulation result of the retrofitted sedimentation tank with a 2 m baffle and sludge hopper.

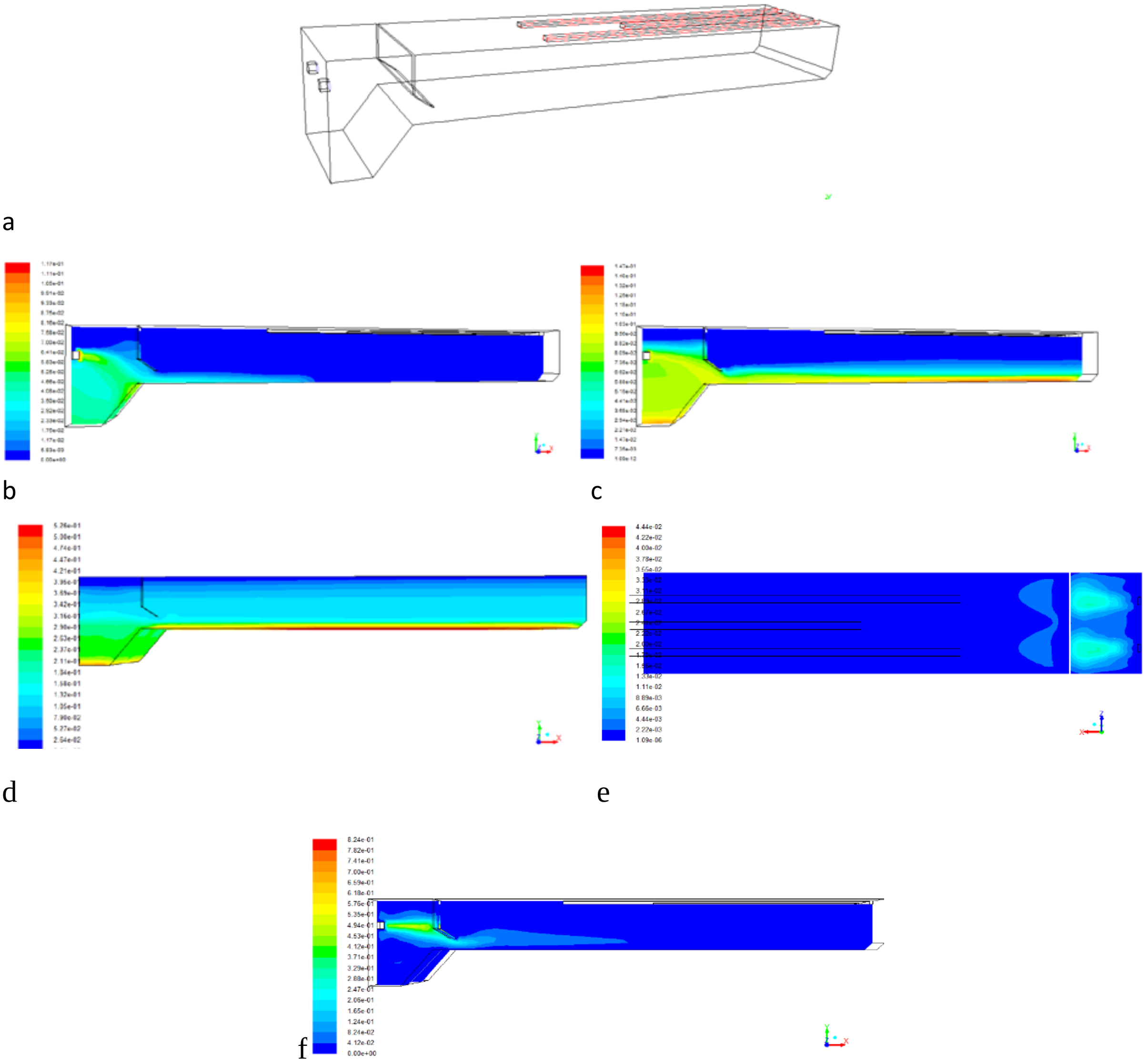
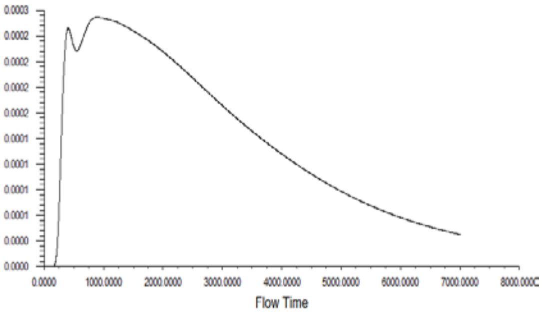


Figure 6 Simulation results of the retrofitted sedimentation tank with a 2 m baffle. Figure 6 (a) – (e) represent distribution of the volume fractions along the tank with time

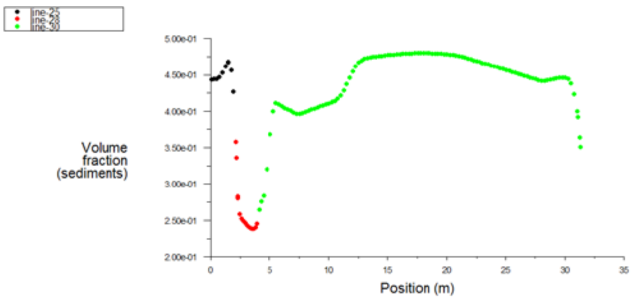
Circulation regions and turbulence intensity in the sedimentation tank could be reduced by using proper baffle in the inlet (Krebs et al. 1995). Goal et.al. (2008) found that an extended baffle provides better influent mixing and isolation between the tank influent and effluent than short baffle, thereby significantly enhancing sedimentation.

We have simulated the retrofitted model of the sedimentation tank with several changes in boundary conditions and shape. This sedimentation tank has the extended, curve baffle at the 2 m away from the inlet, and sludge hoper was presented. Velocity inlet magnitude determine as a boundary conditions too, however, increase velocity magnitude of solids to 1 m/s, velocity magnitude of sewage to 0.8 m/s.

Figure 6 (a) – (d) presents the distribution of the volume fraction during the time, and Figure 6 (e) presents counter of turbulent kinetic energy. As we can see, the biggest part of the particles settle to the sludge hopper, but other part were moving to the outlet (fig.6.i) and turbulent intensity were reduced. But, we got bypass fluid (Fig.6f), its because of new velocity zone apperead between baffle and inlet. First peak reduced on the RTD curve, which meaning the fluid is closed to a “plug glow”.

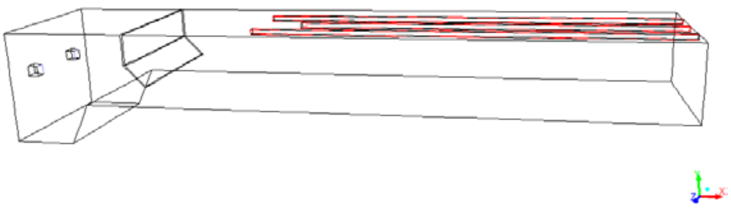


g



i

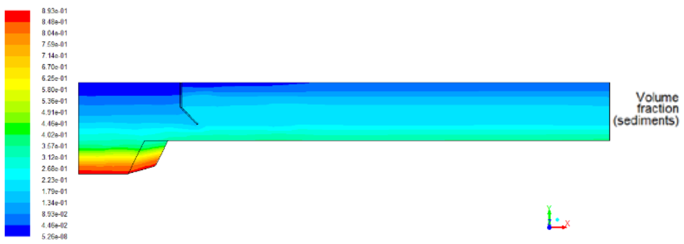
To avoid bypass flow around the baffle, we move baffle to the 4 m, and results are given in the Fig. 7.



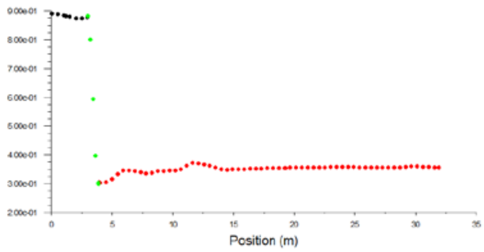
a

b

c



d

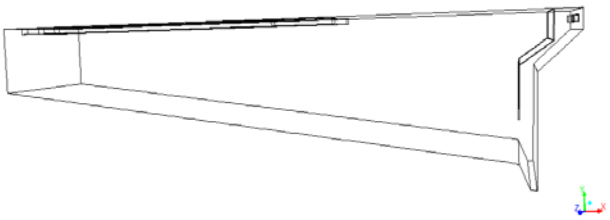


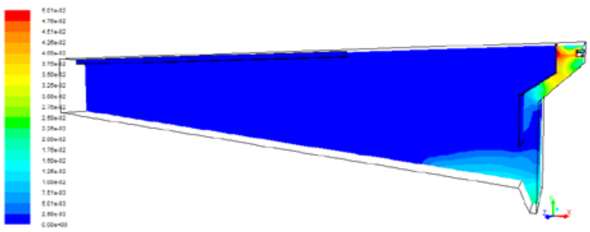
e

Fig.7 (a) – (e) shows that 4 m baffle reduced turbulent intensity, and bypass flow. Volume fraction plot shows that 60 % of the sludge distributed along the bottom, which indicate on the improvements.

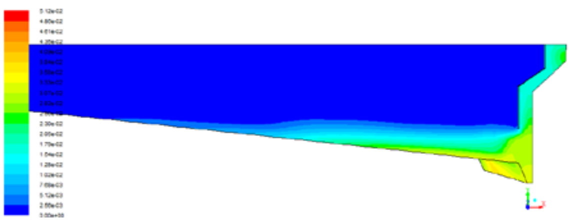
3.3.4 Multi –phase simulation result of the retrofitted sedimentation tank with slopping bottom of 12.5 ▫, and extended curve baffle.

To guide the fluid significantly deeper inside the tank, and to reduce intensity of turbulent kinetic energy, we proposed such retrofitted form of the sedimentation tank.

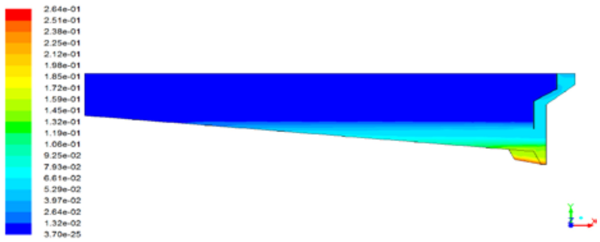




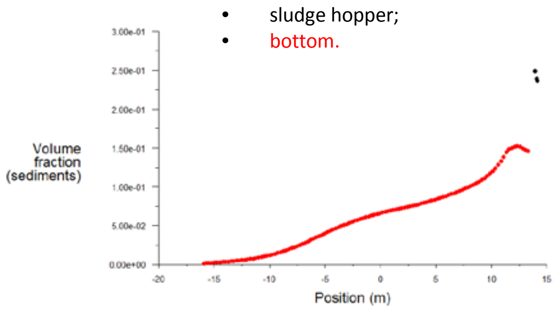
a



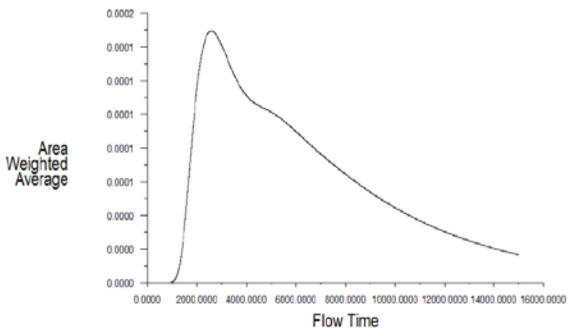
b



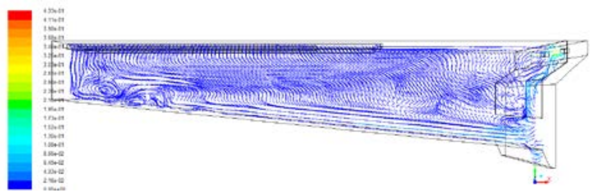
c



d



e



f

Figure 8 Simulation results of the retrofitted model. Distribution of the volume fraction after (a) 100 s, (b) 800 s, (c) 1500 s., (d) plot of volume fraction of sediments. (e) - RTD curve.

Figures 8 (a) – (c) show the evolution of density distribution. The results show that an extended baffle forces the solid to move faster towards the bottom of the tank and decreases the inlet recirculation zone. The RTD plot shows closer results to the ideal “plug flow”. The effect of an extend baffle is also displayed in the Figure 8.d - volume fraction, we can see that 90 % of the sludge in the sludge hopper.

Conclusions

This works deals with the development of the retrofitted model of the treatment of wastewater sedimentation tank. The review of the existing sedimentation tank shows that there are many different hydraulic problems in the existing sedimentation tank. Such as, short retention time, short circuiting, bottom currents, dead zones and bad design. After investigation of the several retrofitted model, we can conclude that baffle strongly increase sedimentation efficiency. Simulation showed that baffle at 2 m and 4 m distance reduce turbulent intensity, circulation zone and bypass flow. The baffle also could push the sludge to the outlet, this problem can be solved by introduction of sludge hoper, which was introduced in the other retrofitted model. To take into account all advantages and disadvantages, the model with slopping 12.5 ▫ bottom, and extended curve baffle was introduced. Extended baffle forces all particles towards the bottom, reduced recirculation zones. RTD curve the closest to the ideal “plug flow”, and distribution of the volume fraction along the bottom show improvements, and increasing of the volume present until 90 %.

In general, CFD can be a powerful tool for the simulation of the sedimentation process, and other water treatment processes. CFD gives us deeper understanding about hydraulic internal processes, helps us to avoid “black – box” effect, tedious, time consuming and expensive experiments. CFD gives new ideas, new possibilities and discover new ways in water treatment process.

References

Anderson, J.D. (1995). *Computational fluid dynamics* (Vol.206). New York: McGraw-Hill. ANSYS Inc. (2009). ANSYS Fluent 14.0 Theory Guide.

Goula, A. M., Kostoglou, M., Karapantsios, T. D., and Zouboulis, A. I.(2008). “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(1–3), 110–121.

Ishii, M. 1975. Thermo-fluid Dynamics Theory of two-phase Flow. Paris: Eyrolles.

Shilton, A., et al. (2010). "An inside look-the potential of CFD technology."

Krebs P., Vischer D., Gujer W. Inlet-structure design for final clarifiers //Journal of environmental engineering. – 1995. – Т. 121. – №. 8. – С. 558-564.

Metcalf, E. (2002). Wastewater engineering: treatment and reuse. *McGraw – Hill, New York.*

Rajvaidya, N., & Markandey, D. K. (1998). *Advances in Environmental Science and Technology. Vol 7:Treatment of Pulp-and Paper Industrial Effluent*. New Delhi: A P H Pub. Corp.

USEPA (2003). "Tracer-Test Planning Using the Efficient Hydrologic Tracer-Test Design (EHTD) Program."

Crowe, C.T., Schwarzkopf, J.D., Sommerfeld and M., Tsuji, Y. (2012): *Multiphase Flows with Droplet and Particles 2nd edition.* CRC Press, Taylor & Francis Group, Boca Raton, Florida.

Thome, J.R. (2004): *Engineering Data Book III,* Wolverine Tube Inc., Decatur, Alabama, USA

Wanot, J., 1996. Computational fluid dynamics methods in ship design. R&D projects (1996), Germany.

Wright, N., and D. Hargreaves. "The use of CFD in the evaluation of UV treatment systems." *Journal of Hydroinformatics* 3 (2001): 59-70.

Amini, R., Taghipour, R., Mirgolbabaei, H. , Numerical assessment of hydrodynamic characteristics in chlorine contact tank. “International Journal for Numerical Methods in Fluids” (2011): 885-898.

Wang H, Falconer RA. Numerical modelling of flow in chlorine disinfection tanks. Journal of Hydraulic Engineering (ASCE) 1998; 124(9):918–931.

Membrane distillation heat transfer enhancement by CFD analysis of internal module geometry

Cipollina, G. Micale, L. Rizzuti Desalination and Water Treatment Vol. 25, Iss. 1-3, 2011

A.Prosperetti and G.Tryggvason (2007). "Computational Methods for Multiphase Flow." Cambridge University Press.

A.Tom (1933). "The Flow Past Circular Cylinders at low Speeds." Proc.Royal Society, A141: 651-666.

Adamsson, Å. (1999). "Computational fluid dynamics for detention tanks."

Andy Shilton, D. G., Paddy Phelps (1999). "An inside look - the potential of CFD thechnology." Water **21**: 37-38.

Cipollina, A., et al. (2011). "Membrane distillation heat transfer enhancement by CFD analysis of internal module geometry." Desalination and Water Treatment **25**(1-3): 195-209.

Cockburn, B., et al. (2000). The development of discontinuous Galerkin methods, Springer.

Glover, G., et al. (2006). "Modelling of wastewater treatment plants how far shall we go with sophisticated modelling tools?" Water Science & Technology **53**(3): 79-89.

Guide, F. U. s. (2006). "FLUENT 6.3 User's Guide."

Huggins, D. L., et al. (2005). "Use of computational fluid dynamics (CFD) for aquaculture raceway design to increase settling effectiveness." Aquacultural engineering **33**(3): 167-180.

Jarvis, P., et al. (2005). "A review of floc strength and breakage." Water Research **39**(14): 3121-3137.

Johnson, F. T., et al. (2005). "Thirty years of development and application of CFD at Boeing Commercial Airplanes, Seattle." Computers & Fluids **34**(10): 1115-1151.

Kostoglou, M., et al. (2006). "Modeling local flotation frequency in a turbulent flow field." Advances in Colloid and Interface Science **122**(1–3): 79-91.

Niyogi, P. (2006). Introduction to computational fluid dynamics, Pearson Education India.

Shilton, A., et al. (2010). "An inside look-the potential of CFD technology."

USEPA (2003). "Tracer-Test Planning Using the Efficient Hydrologic Tracer-Test Design (EHTD) Program."

Versteeg, H. K. and W. Malalasekera (1996). "An Introduction to Computational Fluid Dynamics - The Finite Volume Methid." Longman Scientific, London.