2

A Computational Fluid Dynamics Model of Flow and Settling in Sedimentation Tanks

Ali Hadi Ghawi[1] and Jozef Kriš[2]
[1]*Department of Civil Engineering, Faculty of Civil Engineering, AL-Qadisyia University*
[2]*Department of Sanitary and Environmental Engineering, Faculty of Civil Engineering, Slovak University of Technology, Bratislava*
[1]*Iraq*
[2]*Slovakia*

1. Introduction

Sedimentation is perhaps the oldest and most common water treatment process. The principle of allowing turbid water to settle before it is drunk can be traced back to ancient times. In modern times a proper understanding of sedimentation tank behavior is essential for proper tank design and operation. Generally, sedimentation tanks are characterized by interesting hydrodynamic phenomena, such as density waterfalls, bottom currents and surface return currents, and are also sensitive to temperature fluctuations and wind effects.

On the surface, a sedimentation tank appears to be a simple phase separating device, but down under an intricate balance of forces is present. Many factors clearly affect the capacity and performance of a sedimentation tank: surface and solids loading rates, tank type, solids removal mechanism, inlet design, weir placement and loading rate etc. To account for them, present-day designs are typically oversizing the settling tanks. In that way, designers hope to cope with the poor design that is responsible for undesired and unpredictable system disturbances, which may be of hydraulic, biological or physico-chemical origin.

To improve the design of process equipment while avoiding tedious and time consuming experiments Computational Fluid Dynamics (CFD) calculations have been employed during the last decades. Fluid flow patterns inside process equipment may be predicted by solving the partial differential equations that describe the conservation of mass and momentum. The geometry of sedimentation tanks makes analytical solutions of these equations impossible, so usually numerical solutions are implemented using Computational Fluid Dynamics packages. The advent of fast computers has improved the accessibility of CFD, which appears as an effective tool with great potential. Regarding sedimentation tanks, CFD may be used first for optimizing the design and retrofitting to improve effluent quality and underflow solids concentration. Second, it may increase the basic understanding of internal processes and their interactions. This knowledge can again be used for process optimization. The latter concerns the cost-effectiveness of a validated CFD model where simulation results can be seen as numerical experiments and partly replace expensive field experiments **(Huggins et al. 2005).**

Generally, many researchers have used CFD simulations to describe water flow and solids removal in settling tanks for sewage water treatment. However, works in CFD modelling of sedimentation tanks for potable water treatment, rectangular sedimentation tanks, and iron removal by sedimentation tank in surface and groundwater treatment plants have not been found in the literature. Moreover, the physical characteristics of the flocs may not be such significant parameters in the flow field of sedimentation tanks for potable water, due to the much lower solids concentrations and greater particle size distributions than those encountered in wastewater treatment.

Design of sedimentation tanks for water and wastewater treatment processes are often based on the surface overflow rate of the tank. This design variable is predicated on the assumption of uniform unidirectional flow through the tank. **Dick (1982),** though, showed that many full-scale sedimentation tanks do not follow ideal flow behavior because suspended solids removal in a sedimentation tank was often not a function of the overflow rate. Because of uncertainties in the hydrodynamics of sedimentation tanks, designers typically use safety factors to account for this nonideal flow behavior **(Abdel-Gawad and McCorquodale, 1984).**

It can be concluded from the discussion that the current ways in which STs are designed and modified could and should be improved. Providing a tool that might lead to sedimentation tank optimization, as well as understanding, quantifying and visualizing the major processes dominating the tank performance, are the main goals of this research.

2. Scope and objectives

This research focuses on the development of a CFD Model that can be used as an aid in the design, operation and modification of sedimentation tanks **(Ghawi, 2008).** This model represents in a 2D scheme the major physical processes occurring in STs. However, effect of scrapers and inlet are also included, hence the CFD Model definition. Obviously, such a model can be a powerful tool; it might lead to rectangular sedimentation tanks optimization, developing cost-effective solutions for new sedimentation projects and helping existent sedimentation tanks to reach new-more demanding standards with less expensive modifications. An important benefit is that the model may increase the understanding of the internal processes in sedimentation tanks and their interactions. A major goal is to present a model that can be available to the professionals involved in operation, modification and design of sedimentation tanks.. The ultimate goal of the project is to develop a new CFD methodology for the analysis of the sediment transport for multiple particle sizes in full-scale sedimentation tanks of surface and groundwater potable water treatment plants with high iron concentration. The CFD package FLUENT 6.3.26 was used for the case study of the effect of adding several tank modifications including flocculation baffle, energy dissipation baffles, perforated baffles and relocated effluent launders, were recommend based on their field investigation on the efficiency of solids removal. An overview of the outline of the project is given in Figure 1.

The specific objectives of this research include:

- Improve the operation and performance of horizontal sedimentation tank in Iraq which have been identified as operating poorly, by predicting the existing flow, coagulant dose to remove iron and flocculent concentration distribution of the sedimentation tank by means of CFD techniques.
- Develop a mathematical model for sedimentation tanks in 2D;

- Introduce a flocculation submodel in the general ST model,
- Introduce a temperature submodel in the general ST model.
- Design CFD model for simulation of sedimentation tanks, i.e. grids and numerical descriptions.
- Develop a model calibration procedure, including the calibration of the settling properties, and validate the models with experimental data.
- Evaluate the suitability of CFD as a technique for design and research of rectangular sedimentation tanks for drinking water treatment plants and iron removal.
- Use CFD to investigate the effects of design parameters and operational parameters.

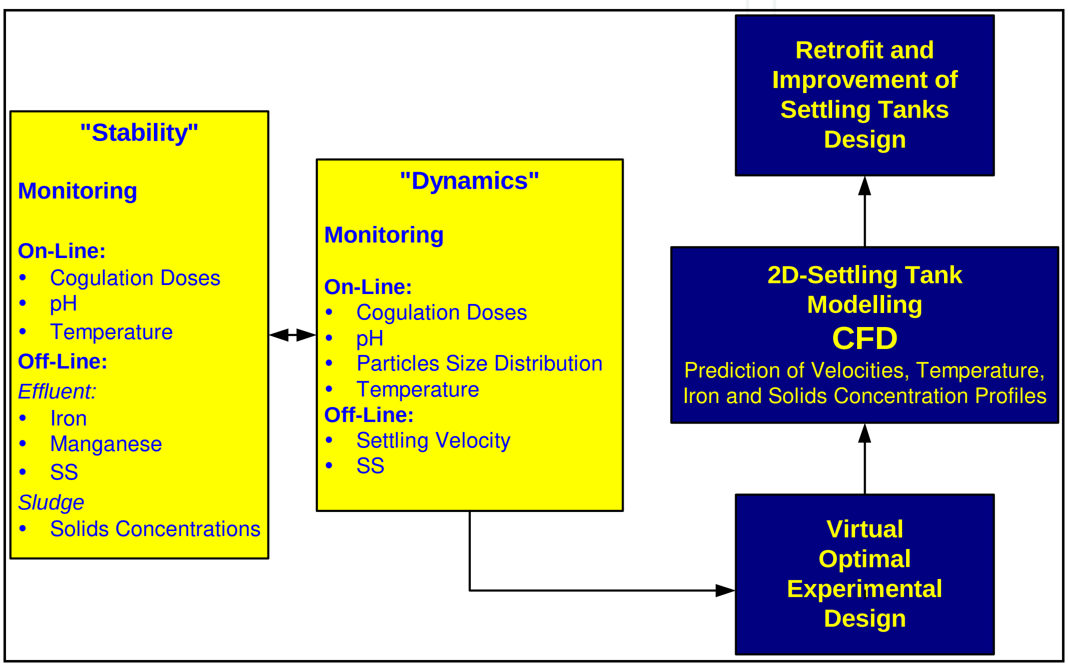


Fig. 1. Overview of the settling tank project

Finally, a CFD model was developed to simulate the full scale rectangular sedimentation tanks at the AL-DEWANYIA purification works in Iraq. The CFD simulations of the AL-DEWANYIA tanks were done by setting up standard cases for each, i.e. a configuration and operating conditions that represented the physical tanks as they were built, and then varying different aspects of the configuration or operating conditions one or two at a time to determine the effect. discrete particles in dilute suspension was simulated, as it is the applicable type for the operating conditions in rectangular sedimentation tanks for potable water treatment.

3. Modelling the settling tank

Figure 2 shows the set-up of the settling tank CFD model which developed in this work. The code predicts fluid flow by numerically solving the partial differential equations, which describe the conservation of mass and momentum. A grid is placed over the flow region of interest and by applying the conservation of mass and momentum over each cell of the grid sequentially discrete equations are derived. In the case of turbulent flows, the conservation equations are solved to obtain time-averaged information. Since the time-averaged equations contain additional terms, which represent the transport of mass and momentum by turbulence, turbulence models that are based on a combination of empiricism and theoretical considerations are introduced to calculate these quantities from details of the mean flow.

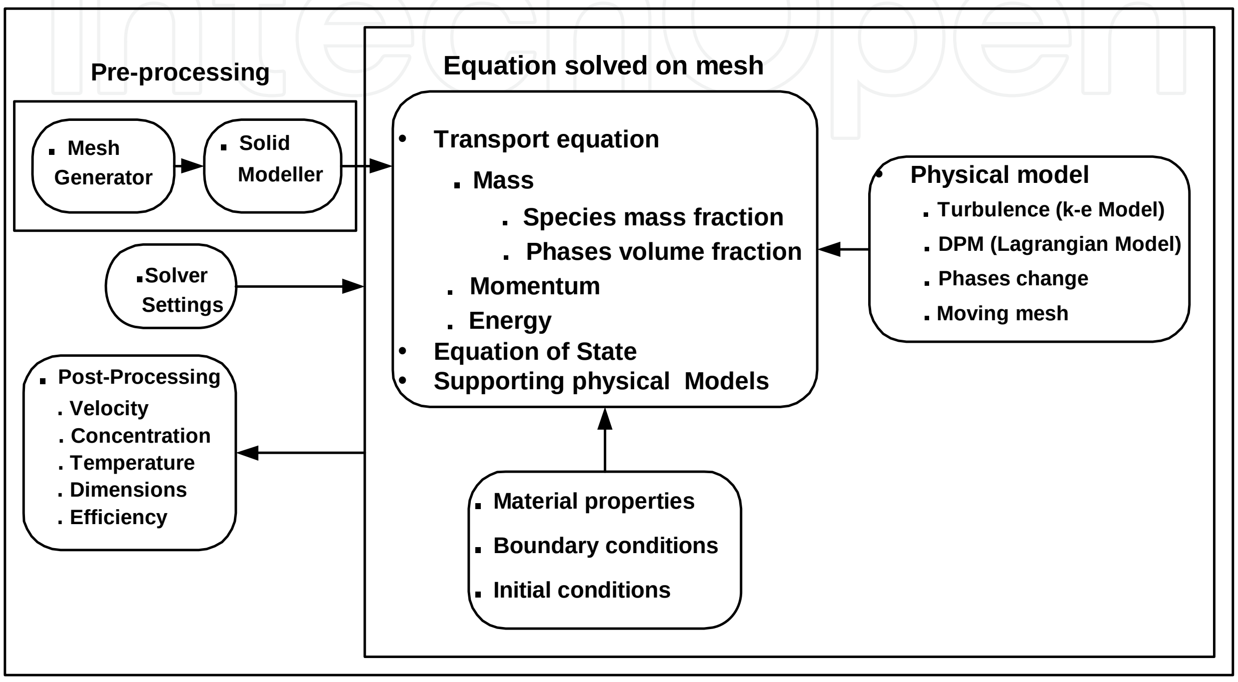


Fig. 2. CFD model

4. Numerical techniques used in Fluent

This section will shortly deal with the methods applied in **(Ghawi, 2008).** The Fluent software utilises the finite volume method to solve the governing integral equations for the conservation of mass and momentum, and (when appropriate) for scalars such as turbulence and solids concentration. In the work (Ghawi, 2008), the so-called segregated solver was applied; its solution procedure is schematically given in Figure 3. Using this approach, the governing equations are solved sequentially, i.e. segregated from one another. Because the governing equations are non-linear (and coupled), several iterations of the solution loop must be performed before a converged solution is obtained.

Concerning the spatial discretisation, the segregated solution algorithm was selected. The k-ε turbulence model was used to account for turbulence, since this model is meant to describe better low Reynolds numbers flows such as the one inside our sedimentation tank. The used discretisation schemes were the simple for the pressure, the PISO for the pressure-velocity coupling and the second order upwind for the momentum, the turbulence energy and the specific dissipation. **Adams and Rodi 1990** pointed out that for real settling tanks the walls can be considered as being smooth due the prevailing low velocities and the correspondingly large viscous layer. Consequently, the standard wall functions as proposed by **Launder and Spalding 1974** were used. The water free surface was modeled as a fixed surface; this plane of symmetry was characterized by zero normal gradients for all variables

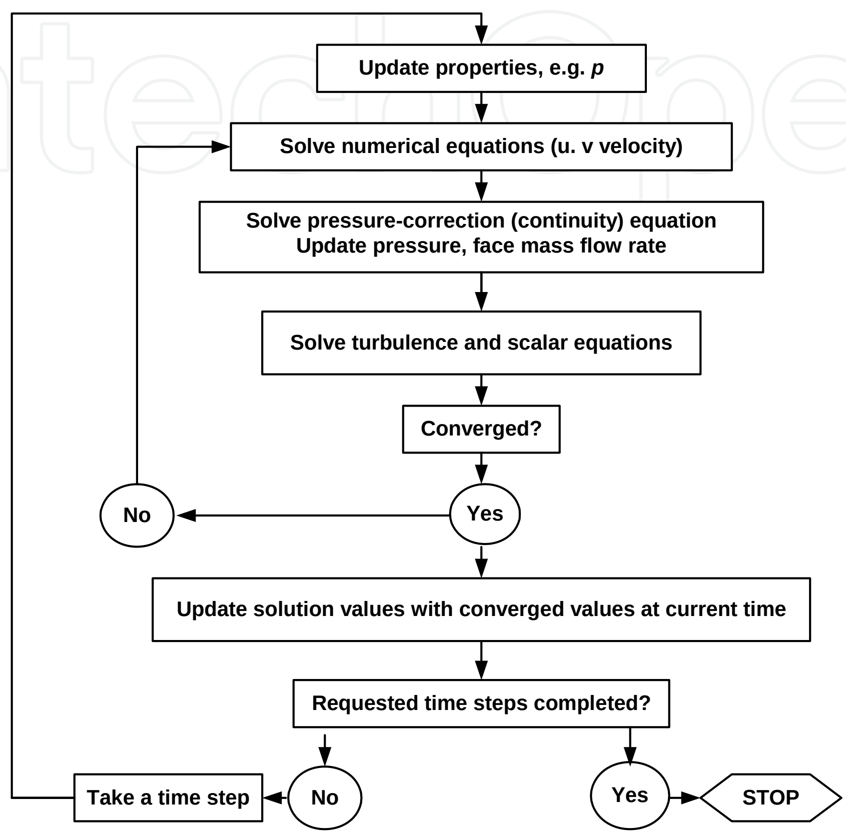


Fig. 3. Solution procedure

5. Experimental techniques for model calibration and validation

The process of developing (incl. calibration), verifying, and validating a CFD code requires the use of experimental, theoretical and computational sciences. This process is a closed loop as presented in Figure 4.

The above clearly indicates that good experimental data are indispensable for settling tank model validation; their quality largely depends on the applied experimental technique.

For the purposes of testing the numerical model presented in this thesis on a full scale tank, the data set gathered laboratories, was selected. Here, a comprehensive experimental study of a working settling tank at AL-DEWANYIA in Iraq were carried out. Velocity and concentration profiles were gathered at 7 stations along the length and 3 stations across the width of the tank for a variety of inlet conditions and inlet and outlet geometries. Volumetric flow rates through the inlets and outlets were measured for each test condition studied. Details of the tank geometry and the experimental conditions for which 3D numerical simulations have been made are given in next sections. The following topics are dealt with which measured in the sites:

Flow rate, (2) Settling velocity, (3) Solids concentration (Turbidity), Iron, and Manganese, (4) Particle size distribution, (5) Velocity of liquid, and (6) Temperature

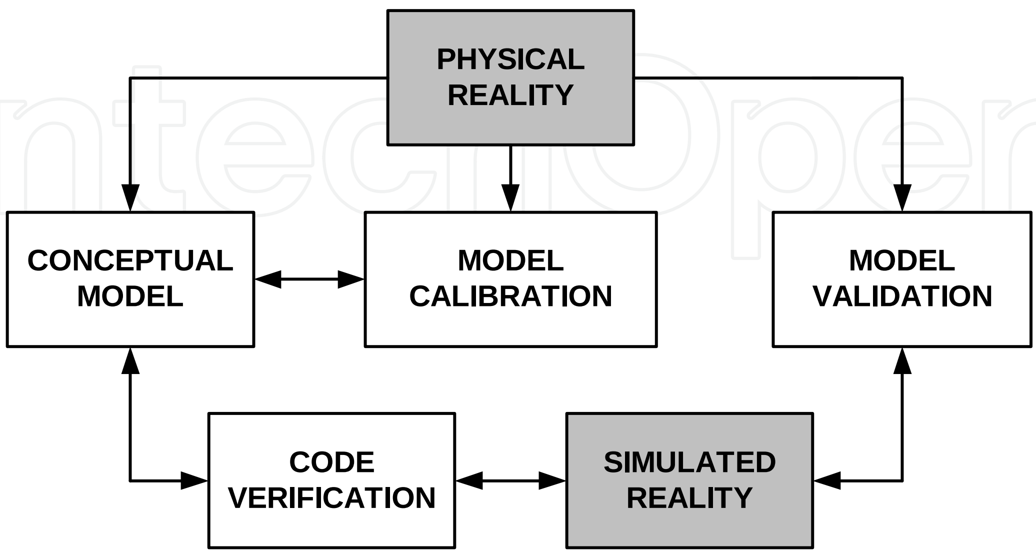


Fig. 4. Process of developing CFD code.

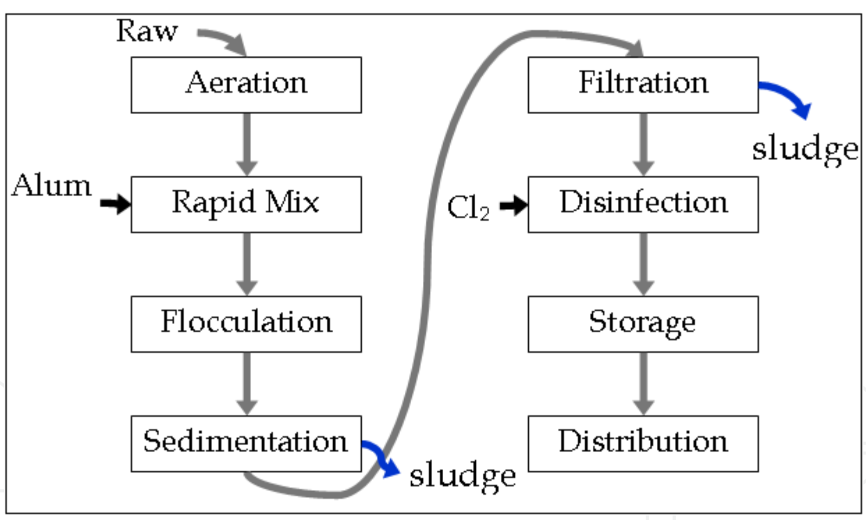


Fig. 5. Layout of AL-DEWANYIA WTP

6. Model development, applications and results

6.1 Introduction

The full-scale horizontal settling tanks at the drinking treatment plant of AL-DEWANYIA were opted for. Most settling tanks of Iraq Water exhibit a horizontal settling tank. This research was focused on this type of settling tanks.

Figure 5 represents treatment of water obtained from a deep well in AL-DEWANYIA WTP a The AL-DEWANYIA WTP were built to remove turbidity and organic material.

6.2 Simulation

To limit computational power requirements, the rectangular settling tank was modeled in 2D. The major assumption in the development of the model is that the flow field is the same for all positions; therefore, a 2D geometry can be used to properly simulate the general features of the hydrodynamic processes in the tank. As a first step, a mesh was generated across the sedimentation tank. As a result, the solutions from the grid of 137,814 quadrilateral elements were considered to be grid independent.

For simulation purposes, the range of the suspended solids was divided into thirteen distinct classes of particles based on the discretization of the measured size distribution. The number of classes was selected in order to combine the solution accuracy with short computing time. Two other numbers, 6 and 15, were tested. While the predictions obtained using 6 classes of particles were found to be different from those resulting from the 13 classes, the difference between the predictions made by the 13 and the 15 classes were insignificant. Therefore, a number of 13 classes were selected as a suitable one. Within each class the particle diameter is assumed to be constant (Table 1). As it can be seen in Table 1, the range of particle size is narrower for classes that are expected to have lower settling rates.

Class	Range of particle size (μm)	Mean particle size (μm)	Mass fraction
1	10-30	20	0.025
2	30-70	50	0.027
3	70-90	80	0.039
4	90-150	120	0.066
5	150-190	170	0.095
6	190-210	200	0.115
7	210-290	250	0.126
8	290-410	350	0.124
9	410-490	450	0.113
10	490-610	550	0.101
11	610-690	650	0.077
12	690-810	750	0.057
13	810-890	850	0.040

Table 1. Classes of particles used to account for the total suspended solids in the STs in AL-DEWANYIA STs.

6.3 The influence of particle structure

The settling velocity of an impermeable spherical particle can be predicted from Stokes' law. However, the aggregates in the water not only are porous but it is well known that they have quite irregular shapes with spatial varying porosity. The flow chart of this computations sequence is presented in Figure 6.

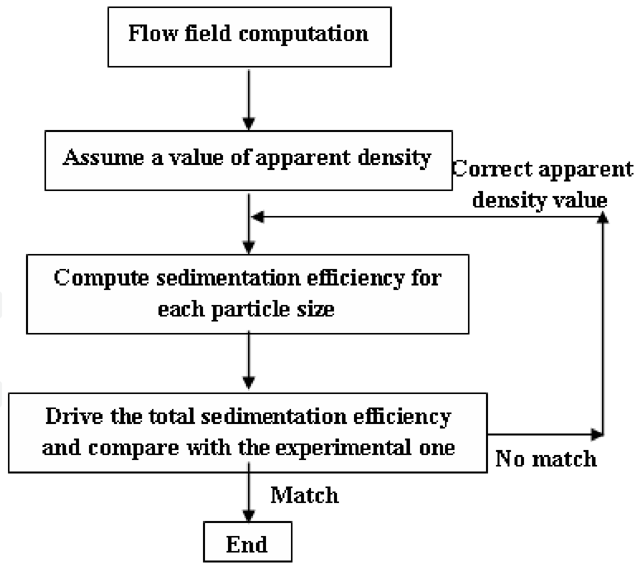


Fig. 6. Flow chart of computation sequence.

6.4 Simulation of existing sedimentation tanks

The AL-DEWANYIA water treatment plant uses lime, and $(NH_4)_2SO_4$ and $Fe_2(SO_4)_3$ to flocculate the and solid concentrations, respectively before entering the sedimentation tanks. There are 4 rectangular tanks at the AL-DEWANYIA WTP .

The Physical and hydraulic data during study periods, and settling tank data for two WTPs are shown in Table 2 .

Geometry	Value
Tank length	30.0 m
Tank width	4.50 m
Hopper depth	2.50 m
Bottom slop	0.00
Weir length	4.50 m
Weir width	0.70 m
Weir depth	0.50 m
loading	Value
SOR	2.7 m/h
Inlet concentration	30-80 mg/l
Density of water	1000 kg/m^3
Density of particulate	1066 kg/m^3
Tank parameter	Value
Average flow rate	60-80 l/s
Sludge pumping rate	5-15 l/s
Inflow temperature average	4^{o}C -11^{o}C , and 20^{o}C -27^{o}C
Inflow suspended solids	25-80 mg/l
Detention time	2.5-3.6 hr
C_{min}	0.17 mg/l
μ	0.002 N.s/m^2

Table 2. Physical and hydraulic data during study periods, and settling tank data.

6.4.1 AL-DEWANYIA WTP

Figure 7 shows the velocity profiles of the existing tanks for a flow rate of 80 l/s and an inlet concentration of 50 mg/l (~75 NTU). High velocities are present at the inlet (0.065 m/s). The flow is further accelerated towards the bottom of the hopper due to the density differences as well as the wedge shape of the hopper. The strong bottom current is balanced by a surface return current inside the hopper. The velocities near the effluent weir are very low.

The solids concentration profile is shown in Figure 8. Note the high concentration downstream of the sludge hopper. The sludge that is supposed to settle in the hopper is washed out of the hopper into the flat section of the tank. Over time a significant amount of sludge accumulates. According to both the field observations and the modeling of the existing process, each of the following reasons (or combination of them) may cause the ST problems, i.e. the flocculant solids blowing out:

1. The location of the existing weir (distributed in a range of 1 meter at the very downstream end of the ST) cause very strong upward currents, which could be one of the major reasons that the flocculant solids were blowing out around the effluent area.
2. The strong upward flow is not only related to the small area the effluent flow passes through but also to the rebound effect between the ST bottom density current and the downstream wall. The "rebound" phenomenon has been observed and reported by many operators as well as field investigators, especially in ST with small amounts of sludge inventory. A reasonable amount of sludge inventory can help dissipate the kinetic energy of the bottom density current.
3. In the existing operation, the bottom density current must be fairly strong due to the lack of proper baffling and the shortage of sludge inventory in the tank.

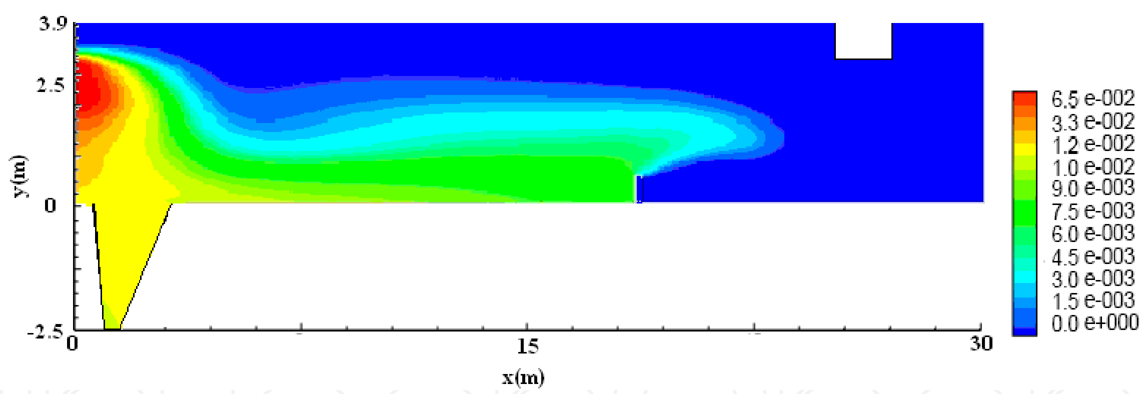


Fig. 7. Velocity contours of existing tank (m/s)

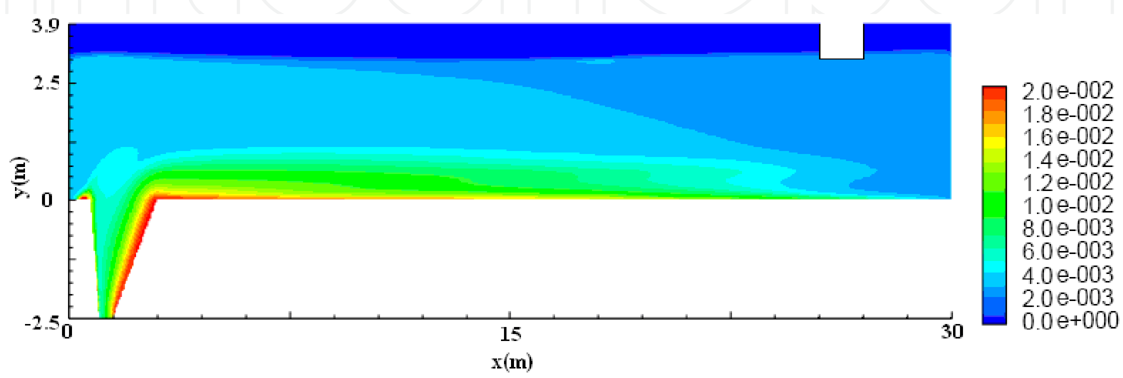


Fig. 8. Solids concentration profile for existing tank

7. Simple improvements to the existing sedimentation tank in WTP

Field data collected from the laboratories during the last 3 years was used to analyze the tank behavior and to enhance the performance of the settling tanks at the AL-DEWANYIA WT Plant. Several tank modifications including flocculation baffle, energy dissipation baffles, perforated baffles and inboard effluent launders, were recommend based on their field investigation

The relationship between the effluent SS and the hydraulic loading is summarised in Table 3 for the existing STs and with different modification combinations. The predicted Effluent SS (ESS) in Table 3 and Figure 9 indicates that the average ESS can be significantly reduced by improving the tank hydraulic efficiency. The comparison of model predictions with the subsequent field data indicates that the significantly improvement of STs performance was obtained by using the minor modifications based on the 2-D computer modeling.

	Q= 50 l/s Influent conc.= 40 mg/l	Q= 70 l/s Influent conc.= 40 mg/l	Q= 80 l/s Influent conc.= 50 mg/l	Q= 80 l/s Influent conc.= 75 mg/l
	Predicted average effluent concentration			
Existing tank	20	30	40	50
Modification 1	12	11	30	22
Modification 2	6	8	12	13
Modification 1 and 2	4	6	7	9
(1) Perforated baffle distance from inlet = 16m; gap above bed = 0.5 m; height above bed = 1.8 m; porosity = 55% (2) Length of launder = 12 m.				

Table 3. Performance data for modelled settling tank

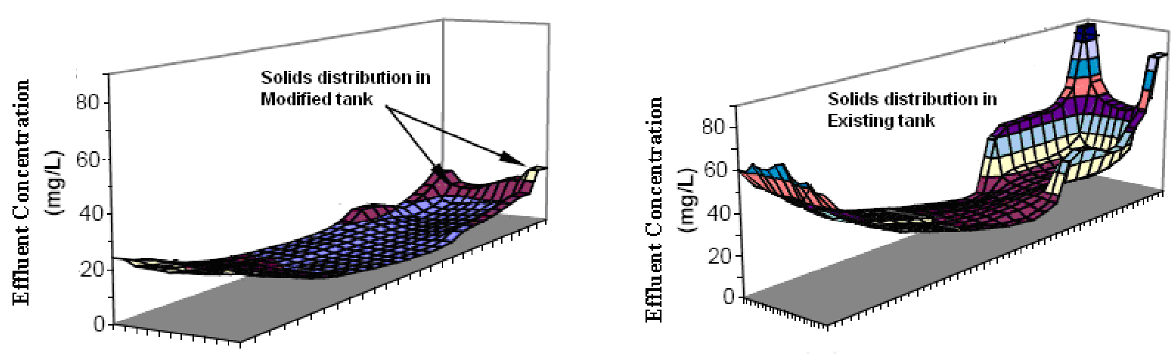


Fig. 9. Comparison of solids distributions on surface layer between existing and modified tanks

8. Modelling the scraper mechanism

The gravitational (and laminar) flow along the bottom, which may go up to 8-15 mm/s near the sump, is blocked for 40 minutes of scraper passage. This is clearly seen in Figure 10. The scraper blade thus constrains the bottom flow discharge by counteracting the gravitational force. Near the floor the velocity increases with height in the shear flow region, but is obviously limited by the scraper's velocity.

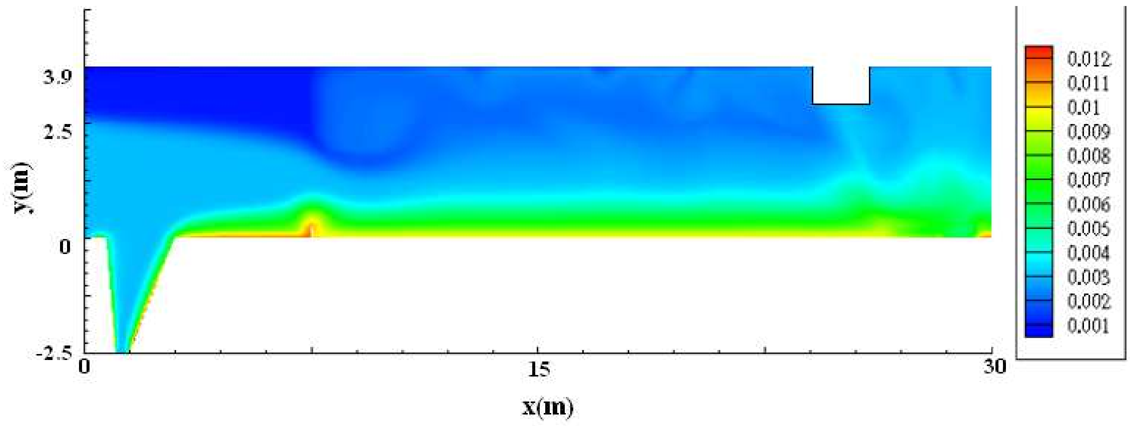


Fig. 10. Effect of scraper on solids concentration profiles

9. Design procedures and guidelines

The design procedures are necessarily based on many assumptions, not normally stated as shortcomings and limitations during the design process. To demonstrate the implications of these assumptions and the way in which these assumption deviate from real tanks. **(Ghawi, 2008)** tried to improve design procedure as show in Table 4.

Improved design procedure	
Step	description
Step 1	Measurement of settling velocity and sludge density
Step 2	Set up of computational grid
Step 3	Simulate tank
Step 4	Evaluate results and check for evidence of the following: - short circuiting - high velocities zones - high overflow concentration - poor sludge removal
Step 5	If none of the above is present, tank size can be reduced to reduce capital cost. If problems are evident, adjust the design by adjusting the: - inlet - position of sludge withdrawal - position of overflow launders Also consider using perforated, porous and deflecting baffles
Step 6	Repeat until a satisfactory tank geometry is obtained and check final geometry for various process changes such as density, concentration and inflow rate.
Step 7	Asses the influence of the settling velocity and sludge density input parameters and repeat steps 3 – 6 if necessary.

Table 4. Proposed CFD enhanced design procedure.

10. Temperature effect

Settling velocity correction factor

In order to define a correction factor for the settling velocities based on temperature difference, the temperature effect on the zone settling velocity has to be determined.
Figure 11 displays graphically the value of the relationship Vs_{T2} / Vs_{T1} and μ_{T2} / μ_{T1} for the data presented in Table 5 at temperatures T_s (summer temperature) and T_w (winter temperature).

CFD Calculated at summer temperature				
SS mg/l	**Settling velocity V m/h**	**Inlet temperature °C**	**Outlet temperature °C**	**Dynamic viscosity μ kg/m.s**
60	1.5	27.5	27.5	8.5e-04
50	1.7	27.5	27.5	8.6e-04
25	1.83	26	26	8.7e-04
15	2.52	25.4	25.4	8.8e-04
CFD Calculated at cooled temperature				
60	0.95	8	9.2	1.3e-03
50	1.05	6.6	6.8	1.35e-03
25	1.9	7.8	8.8	1.29e-03
15	2.7	7	8.9	1.30e-03

Table 5. Settling velocity and dynamic viscosities for summer and winter temperature.

From Figure 11 can be observed that the numerical values of the ratios Vs_{T2} / Vs_{T1} and μ_{T2} / μ_{T1} are very close, suggesting that an easy correction in the zone settling velocity for different temperatures can be made with a correction factor based on the dynamic viscosity of the water at the two temperatures. Figure 12 shows an extended data set indicating the relationships between the ratios Vs_{T2} / Vs_{T1} and μ_{T2} / μ_{T1}.
Fitting a straight line to the data point presented in Figure 13 can find a correction factor for the settling velocities based on temperature

$$V_{ST_2} = V_{ST_1}\left(\frac{10^{\left[\frac{247.8}{T1+133.15}\right]}}{10^{\left[\frac{247.8}{T2+133.15}\right]}}\right) \tag{1}$$

Equation 1 can be applied to correct the settling velocities for difference in temperatures in whichever of the four types of sedimentation, i.e., unflocculated discrete settling, and flocculated discrete settling. Even though equation 1 can be used for a sensitivity analysis on the performance of the model for different seasons, e.g. summer and winter, there is no evidence that the settling properties can be accurately extrapolated from one season to another.

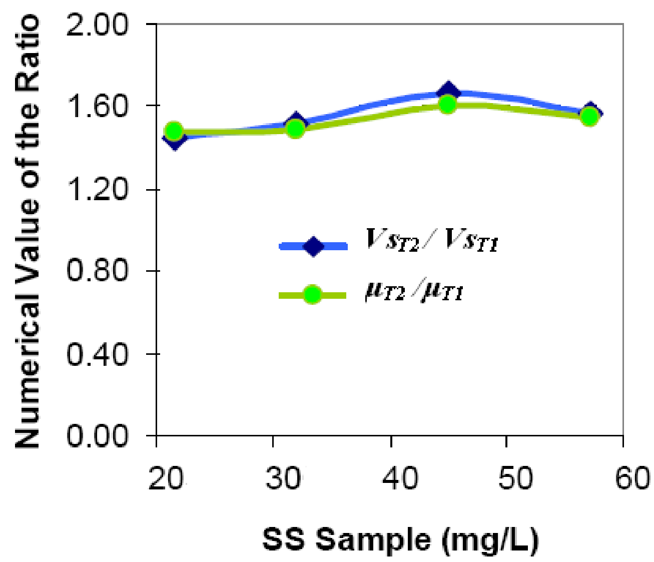


Fig. 11. Ratios of Vs_{T2} / Vs_{T1} and μ_{T2} / μ_{T1} for Different suspended solid (SS) concentrations.

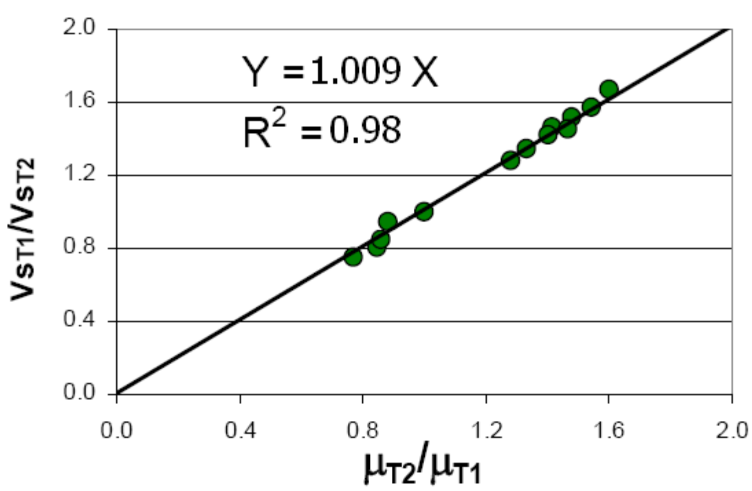


Fig. 12. Effect of Temperature on Settling Velocity.

11. Validation of the model

The validation process involves comparing the model response to actual measured data. The model was validated using measured data from the AL-DEWANYIA WTPs.

After the development of the hydrodynamic model, and turbulence model, the ST model was tested. The ESS predicted by the model was tested during seven days (from a 10 day period) showing a very good agreement with the field data. Figure 13 presents a comparison between the experimentally measured and the simulated values of the floc concentration in the effluent of the existing tanks in AL-DEWANYIA. Apparently, there is a good agreement between measured and predicted values.

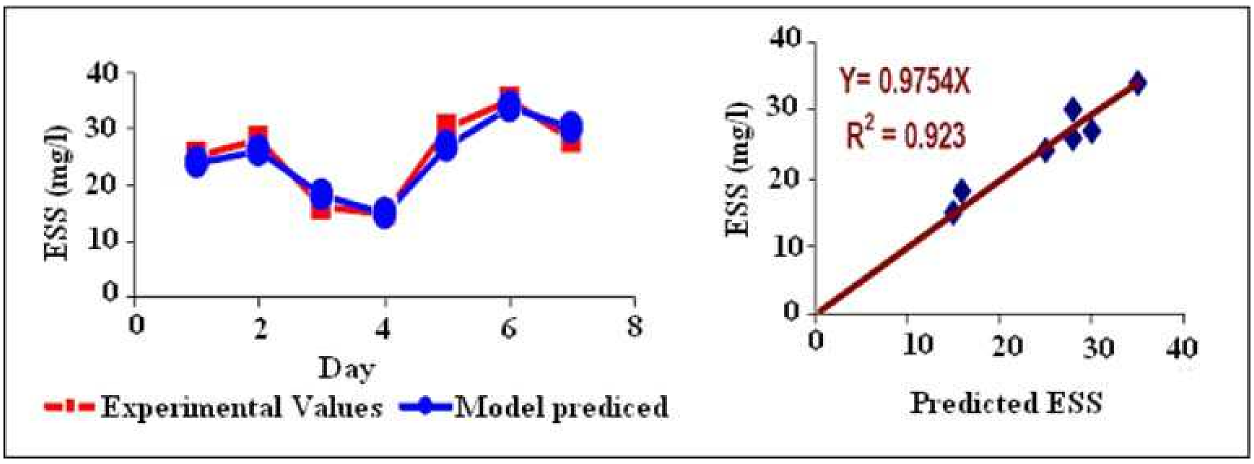


Fig. 13. Validation of the ESS Simulated by the Model

The average values of effluent concentration of improvement are presented in Table 6. The tank operation conditions in the data collection periods (February-April, 2007, (June-August, 2006)) and the CFD model predictions are very close as shown in Table 6. The comparison of model predictions with the subsequent field data indicates that the significantly improvement of tank performance was obtained by using the minor modifications based on the 2-D computer modeling.

	Operation conditions	Effluent concentration (mg/l) and improvement		
	Ave. concentration mg/l	No Modifications	baffle Modifications	Baffle and launder Modifications
Field Data June-August 2006	50	28	-	-
Model Predictions	47	27	6 (+78%)	5 (+82%)

Table 6. Comparison of model predictions with field data

12. Conclusions

The introduction of this study made clear that many factors influence the performance of settling tanks. They may be categorised as physico-chemical and hydraulic influences. To account for them in terms of process operation and design, mathematical models may be utilised. In this respect, Computational Fluid Dynamics (CFD) enables the investigation of internal processes, such as local velocities and solids concentrations, to identify process in efficiencies and resolve them. Although these complex models demand for considerable computational power, they may become an option for the study of process operation and control as computer speed increases. Nowadays, they mostly find applications in the world of settling tank design.

The main purpose of this investigation was to develop a CFD ST model capable of simulating the major processes that control the performance of settling tanks, this goal was achieved. The accomplished objectives of this research include: the development of a compound settling model that includes the representation of the settling velocity for the suspended solids usually encountered in this type of tank (horizontal sedimentation tanks) the inclusion of iron removal effects, a flocculation sub-model, and a temperature sub-model.

These types of sub-models have not been previously incorporated in CFD ST models. The model was rigorously tested and validated. The validation process confirms the utilities and accuracy of the model. An important benefit of this research is that it has contributed to a better understanding of the processes in STs. The results presented in this research clarify important points that have been debated by previous researchers.

This research may also open the discussion for future research and different ways for improving the performance of existing and new STs. In summary, this research has led to more complete understanding of the processes affecting the performance of settling tanks, and provides a useful tool for the optimization of these corn stone units in water treatment. The major conclusions, general and specifics, obtained from this research are:

1. CFD modeling was successfully used to evaluate the performance of settling tank.
2. The usually unknown and difficult to be measured particle density is found by matching the theoretical to the easily measured experimental total settling efficiency. The proposed strategy is computationally much more efficient than the corresponding strategies used for the simulation of wastewater treatment.
3. Solid removal efficiency can be estimated by calculating solids concentration at effluent.
4. High solid removal efficiency was achieved for all cases tested.
5. Baffling inlet arrangement succeeded in controlling kinetic energy decay.
6. Improved energy dissipation due to an improved inlet configuration.
7. Reduced density currents due to an improved inlet configuration.
8. Improved sludge removal due to the inlet configuration.
9. Troubleshoot existing STs and related process operations.
10. The effluent quality can be improved by more than 60% for any cases.
11. Evaluated ST design under the specified process conditions.
12. Develop reliable retrofit alternatives with the best cost-effectiveness.
13. The changes in temperature on STs play an important role on the performance of STs.
14. Scrape is important in the settling process and play a big role in changing the flow field.
15. In this work we improved the STs guidelines design procedure.
16. The fairly good agreement between model predictions and field data.

In general the study demonstrated that CFD could be used in reviewing settling tank design or performance and that the results give valuable insight into how the tanks are working. It can be inferred that CFD could be use to evaluate settling tank designs where the tanks are not functioning properly.

13. Acknowledgement

This chapter is supported by the Scientfic Grant Agency of Ministry of Education-VEGA Projekt No. 1/1143/11 with at the Department of Sanitary and Environmental Engineering, Faculty of Civil Engineering, Slovak University of Technology in Bratislava.

14. References

[1] Huggins D.L., Piedrahita R.H., & Rumsey T., (2005). Use of computational fluid dynamics (CFD) for aquaculture raceway design to increase settling effectiveness, Aquacult. Eng. 33, 167–180.

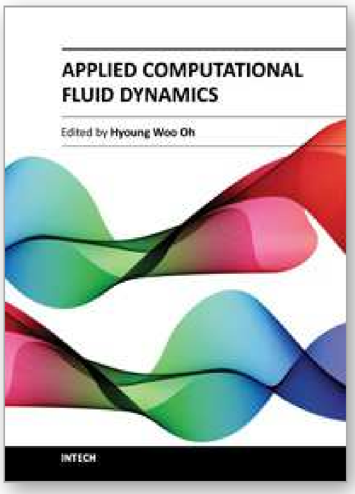
[2] Dick, R. I. (1982) Sedimentation Since Camp, J Society of Civil Engrs, 68, 199-235.

[3] Abdel-Gawad, S. M. & McCorquodale, J. A. (1984b). Hydrodynamics of Circular primary clarifiers. Canadian J. Civil Engr. ,11, 299-307.

[4] Adams, E.W. & Rodi, W. (1990). Modelling flow and mixing in sedimentation tanks, Journal of Hydraulic Engineering, 116(7):895-913, Jul.

[5] Launder, M. & Spalding, D. (1974). The numerical computation of turbulent flows, Computer methods in applied mechanics and engineering, (3), 269-289.

[6] Ghawi, A.H., (2008). A numerical model of flow and settling in sedimentation tanks in potable water treatment plants, Edícia vedeckých prác (Science Works Edition) , Workbook no. 64, STU Bratislava, pp. 150, ISBN 978-80-227-2964-2



Applied Computational Fluid Dynamics

Edited by Prof. Hyoung Woo Oh

ISBN 978-953-51-0271-7
Hard cover, 344 pages
Publisher InTech
Published online 14, March, 2012
Published in print edition March, 2012