Ali Hadi GHAWI, Jozef KRIŠ

Slovak University of Technology , Faculty of Civil Engineering,
Department of Sanitary and Environmental Engineering

DESIGN AND OPTIMIZATION OF SETTLING TANKS PERFORMANCES IN SLOVAKIA

KONSTRUKCJA ORAZ OPTYMALIZACJA FUNKCJONOWANIA ZBIORNIKÓW SEDYMENTACYJNYCH NA SŁOWACJI

A model of the rectangular settling tanks proposed for the Water Treatment Plant of Holic in Slovakia was prepared using Computational Fluid Dynamics (CFD). The modeling approach used was two dimensional, multi-phase simulations with solids transport and removal included, and reflected the state of the art in sedimentation modelling. Special attention was paid to the inlet baffles and outlet weirs in the model setup. The techniques used within and the general results of the model will be presented. The results include effluent solids concentrations, and solids concentration profiles and velocity magnitude contours for various tank modifications compared to the existing tank. 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.

Model prostokątnych zbiorników sedymentacyjnych zaproponowany dla Stacji uzdatniania wody w Holic (Słowacja) został przygotowany w oparciu o technologię Komputerowej mechaniki płynów. Zastosowana metoda projektowania to dwuwymiarowe, wieloetapowe symulacje z uwzględnieniem transportu i odprowadzania substancji stałych. Odzwierciedla ona nowoczesne dokonania w zakresie projektowania zbiorników sedymentacyjnych. W konstrukcji modelu szczególną uwagę zwrócono na przegrody wlotowe i przelewy wylotowe. Zostaną przedstawione zastosowane techniki oraz ogólne wyniki funkcjonowania modelu. Wyniki obejmują stężenia dopływowe substancji stałych oraz profile stężeń substancji stałych oraz wykresy wartości prędkości dla różnych wersji zmodyfikowanych zbiornika w porównaniu do istniejącego zbiornika. Podsumowując, badanie wykazało, że technologia CFD może być stosowana podczas rozpatrywania konstrukcji lub wydajności zbiornika oraz że wyniki dają wgląd w sposób ich funkcjonowania. Można wywnioskować, że technologia CFD może być stosowana do oceny konstrukcji zbiorników sedymentacyjnych w przypadku ich nieprawidłowego funkcjonowania.

1. Introduction

Groundwater is an important source of municipal drinking water for many small and medium-sized communities in Slovakia. Many favour groundwater over surface water because of its excellent and consistent quality, and because, generally, it requires little or no treatment before consumption. Unfortunately, many groundwater supplies are contaminated by varying levels of iron and manganese in concentrations that exceed the Slovakian Drinking Water Guidelines.

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 physic-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).**

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 groundwater treatment plants have not been found in the literature

Figure 1 represents treatment of water obtained from a deep well (groundwater) in Holic WTP. The Holic WTP were built to remove iron manganese, turbidity and organic material.

The aim of this project was originally stated as to improve the operation and performance of horizontal sedimentation tank in Hrinova water treatment plants which have been identified as operating poorly by means of CFD techniques. Also the main objective of the study was the comparison of the different inlet and outlet arrangements. To accomplish this it was felt that a three dimensional, multi-phase model of the settling tanks would be necessary. A model of this type would permit that many secondary objectives could also be achieved. Secondary objectives include being able to study in detail the hydrodynamics of the settling tanks, visualizing the effects of the inlet and outlet arrangements as well as flow path lines, treated water and activated sludge. To achieve these secondary objectives, which really guarantee that the main objective is achieved, many process and CFD issues needed to be addressed.

Principal among these issues are the following:

- Modeling of inlet baffles (energy dissipating and distribution)
- Modeling of solids settling and concentration
- Modeling of sludge transport and removal

How each of these issues was resolved is the main theme of this paper.

2. Methodology

The FLUENT Computational Fluid Dynamics (CFD) code was used to calculate the flow field. The Computational Fluid Dynamics (CFD) model was used to calculate the flow field. Multiphase applications generally require the user to provide information on the drag interaction between phases. This phase-to-phase characterization is key to an appropriate process description and usually involves experimental work and additional parameter tuning after running the model. Alternatively, a Boussinesq approach was used in the present simulations, which reduces complexity by waiving the drag coefficient determination. In the sedimentation model an additional scalar equation was added to include the concentration of the solids. The k-ε model was used in this study.

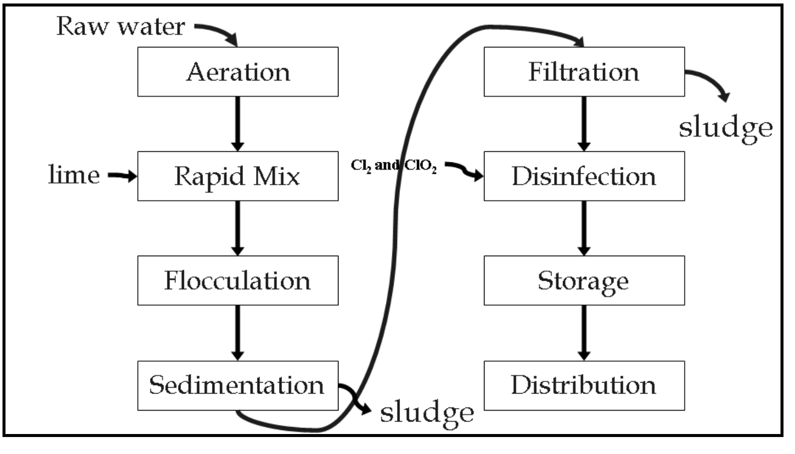


Fig. 1 Layout of Holic WTP

Rys. 1. Układ — Holic WTP

Fig. 2 Picture of Holic sedimentation tank

Rys. 2. Ilustracja — zbiornik sedymentacyjny Holic

3. Material and Methods

The numerical model is compared with data obtained in a settling tank. In this study, mean velocity and solids fraction concentration were measured at a number of stations along the length and across the width of the Holíc settling tanks.

The time taken to gather all the data for one inlet condition was around 3 hours, so no distinct time was given for the measurements at each station. Because of this, the numerical simulation was compared with the experimental data at a time when the settled sludge layer in the simulation was at approximately the same height as that found in the experiment. At this time, the flow field above the settled sludge layer should be similar in both the experiment and the simulation.

Tab. 1 Physical and hydraulic data during study periods, and settling tank data.

Tab. 1. Dane fizyczne i hydrauliczne podczas okresów badawczych oraz dane dotyczące zbiornika sedymentacyjnego.

Geometry	Value
Tank length	30.0 m
Tank width	4.50 m
Water depth	3.80 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
Inlet concentration Density of water Density of particulate	80 mg/l 1000 kg/m^3 1066 kg/m^3
Tank parameter	**Value**
Average flow rate	80 l/s
Sludge pumping rate	5 l/s
Inflow temperature,	5^{o}C -11^{o}C , and 25^{o}C -27^{o}C
Inflow suspended solids	25-80 mg/l
Detention time	3.6 hr
Minimum concentration	0.17 mg/l
μ	0.002 N.s/m^2
No. of Tank	2

The horizontal settling tank is shown in Figures 2**,** it is 30 m long, 4.5 m wide and has a maximum depth at the hopper of 2.5 m. Longitudinal velocity was determined by means of sensors, a method that involves measuring the time taken for a drifter at the measurement depth to traverse a given distance from which the velocity could be calculated. Accuracy is given to +/- 2mm/s.

Solids concentration was measured by gravimetric determination method and sensor. Interference with concentration measurements occurred when the probe was placed-within 0.2 m of the walls or sludge scraper, hence no readings were taken closer than this distance. Also these measurements were limited to an upper solids fraction of about 0.07, around 1.3 times the inlet solids fraction.

Tank Inlet. Each tank is fed by pipe from flocculation tank. As the water enters the sedimentation tank. This arrangement is shown in Figure 3. One pipe 0.8 m diameter allows the flocs to leave the flocculation tank and enter into the settling tank (see Figure 3).

Tank Outlets. Sludge is withdrawn from the base of the sludge hoppers by a pump. The clear water is withdrawn from the tank surface by effluent outlet weir located at 29m, along the width of the tank (Table 1).

Sludge Scrapers. Settled sludge is moved along the floor of the tank towards the sludge hopper by a continuous chain scraper. This scraper moves baffles 0.125m deep at a rate of 0.008 m/s towards the inlet.

Inlet Conditions. The total inlet volumetric flow rate was 80 l/s.

The density difference between inlet and effluent is between 1.05-2 kg/m^3 for all the experiments. The sludge density is 1066 kg/m^3, which is the density of the hydrated sludge flocs in suspension. This is low compared to dry sludge densities previously reported between 1066 and 2000 kg/m^3, **Larsen (1977) and Dahl (1993).**

This is rather high in comparison to the inlet solids fraction reported in the previous simulations.

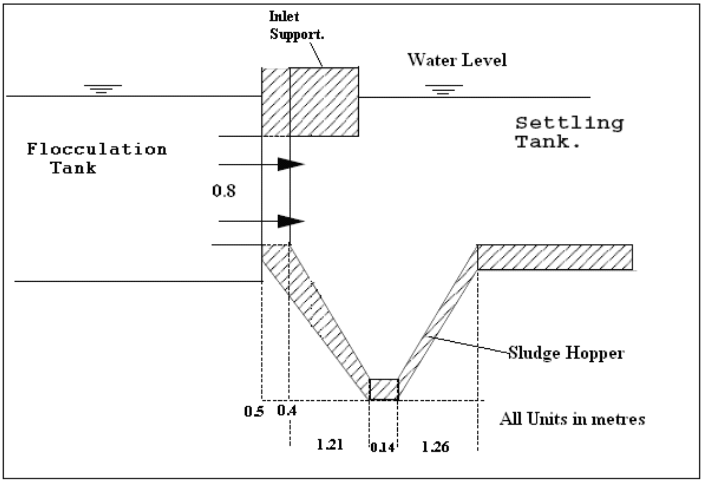


Fig. 3 *Schematic representations of the Holic sedimentation tank inlet and hopper.*

Rys. 3. *Schemat zbiornika sedymentacyjnego Holic — wlot oraz lej*

A rectangular sedimentation tank in Slovakia water treatment plants was selected to demonstrate the response of rectangular tanks to different internal geometries. This case is based on the Holic settling tanks describe in this study. This tank was selected because performance data are available for model calibration, and because it represents a marginal performance case. Table 1 shows the main tank dimensions and loadings.

The model hydrodynamic parameters were calibrated before the model was applied. The data on velocities and concentrations were used to adjust the model bed roughness and vertical Prandtl-Schmidt number.

4. Results and Discussion

A rectangular sedimentation tank in Slovakia water treatment plants was selected to demonstrate the response of rectangular tanks to different internal geometries. This case is based on the Holic settling tanks describe by **Ghawi and Kris (2007 a) and Ghawi and Kris (2007 b).** This tank was selected because performance data are available for model calibration, and because it represents a marginal performance case. **Table 1** shows the main tank dimensions and loadings. Figure 4 and 5 shows an idealized profile of the Holic Settling Tank (ST).

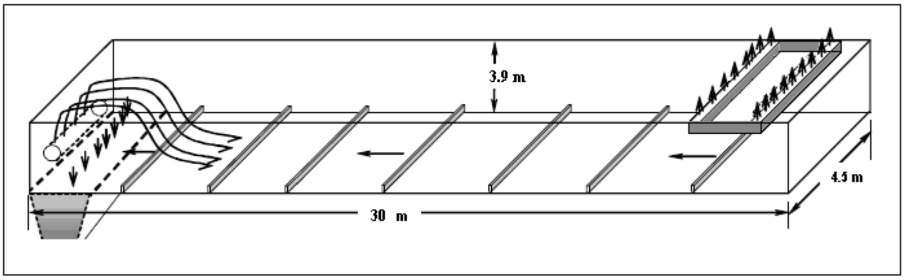


Fig. 4 Schematic representation of the Holic sedimentation tank in 3D.

Rys. 4. Schemat zbiornika sedymentacyjnego Holic — 3D

Fig. 5 Picture for Holic settling tank

Rys. 5. Zdjęcie zbiornika sedymentacyjnego Holic

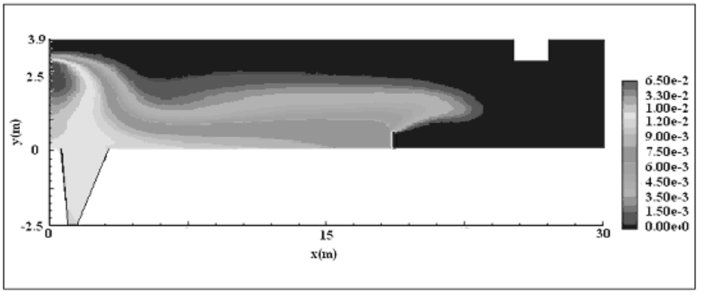


Fig. 6 Velocity contours of existing tank (m/s)

Rys. 6. Wykresy prędkości w istniejącym zbiorniku (m/s)

Figure 6 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.

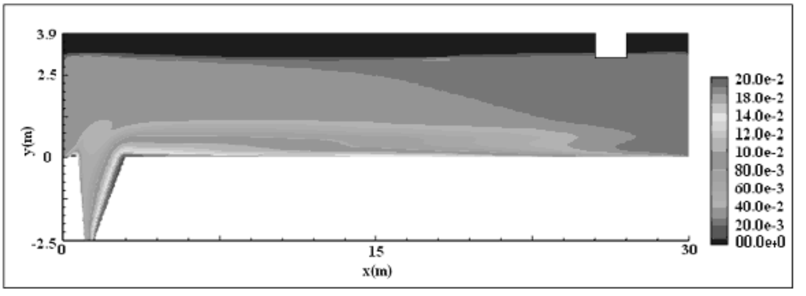


Fig. 7 *Solids concentration profile for existing tank*

Rys. 7. *Profil zagęszczenia cząstek stałych dla istniejącego zbiornika*

The solids concentration profile is shown in Figure 7. 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. Figure 8 clearly shows the location and the extent of the accumulated settled sludge downstream of the sludge hopper.

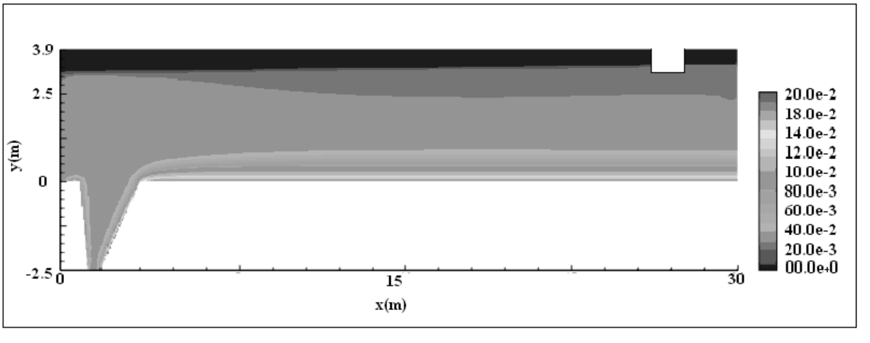


Fig. 8 *Solids concentration profile for existing tank*

Rys. 8. *Profil zagęszczenia cząstek stałych dla istniejącego zbiornika*

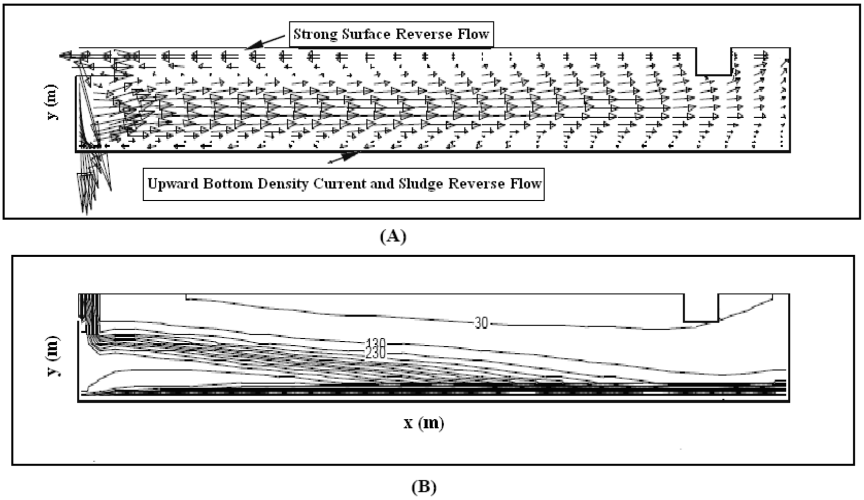


Fig. 9 *Flow Pattern (A) and Sludge Blanket (B) (in a vertical section along tank central axis) in Existing Tank*

Rys.9. *Charakterystyka przepływu (A) oraz kożuch osadu (B) (w przekroju pionowym wzdłuż środkowej osi zbiornika) w istniejącym zbiorniku*

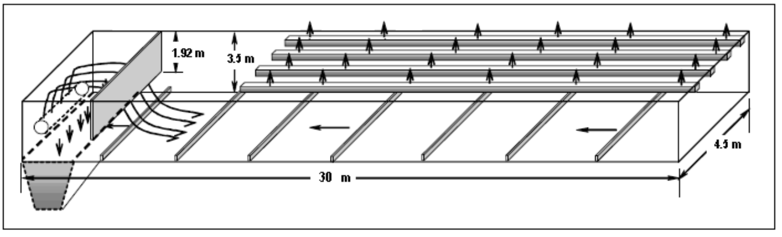
As shown in Figure 9 (a) and (b), the predicted hydraulic regime typically consists of the upward inlet jet; the influent density waterfall, a bottom density current and a strong surface reverse flow in the absence of proper baffling. For a case with a thick sludge blanket, the simulated velocity field showed that the bottom density current deflects upward while near the tank bottom a strong reverse sludge flow appears. According to both the field observations and the modelling 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.

5. Proposed Tank Modifications

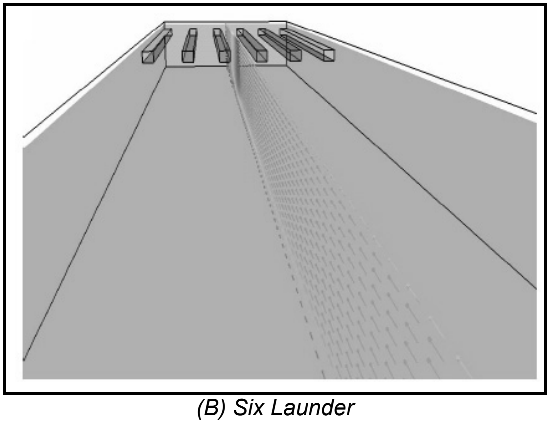
The proposed modifications include: energy dissipation baffles, a flocculation zone, perforated and non-perforated baffles, modifications to the launders and modifications of sludge withdrawal facility. In the original project, total 6 of proposed modifications and combinations of them were tested. The following major modifications adopted in final tank construction are presented as:

- Modification 1 Inlet flocculation baffle, the distance from tank influent to the baffle = 6 m and the baffle depth = 2.6 m;
- Modification 2 A perforated baffle between bay A and B with slot space of 52% of flow cross section area;
- Modification 3 A perforated baffle between bay B and C with slot space of 65% of flow cross section area;
- Modification 4 a conventional baffle between bay A and B with baffle depth of 1.73 m below the surface.
- Modification 5 a conventional baffle between bay B and C with baffle depth of 1.39 m below the surface.
- Modification 6 Removing existing surrounding effluent weir and adding 4 to 6 new launders in the bay C (Figure 10 A and B). All effluent launders are aligned with the tank longitudinal direction. The launders extend from the end wall to the perforated baffle between B and C.



(A) Four Launder

(A) Cztery płuczki

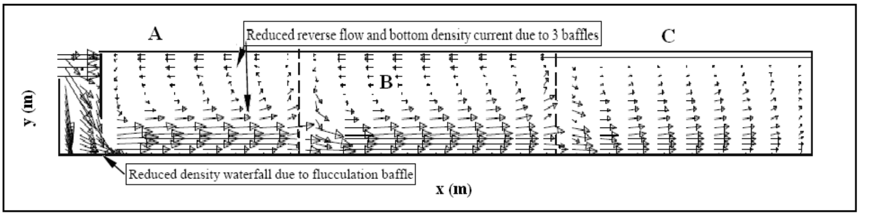


(B) Six Launder

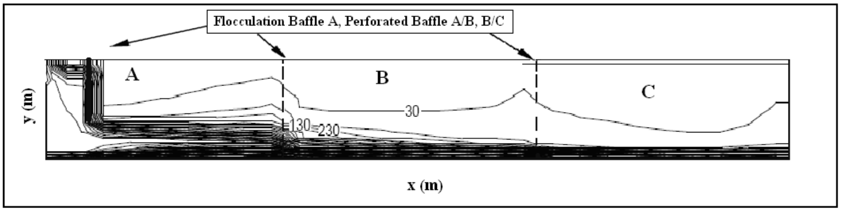
(B) Sześć płuczek

Fig. 10 Tank with baffle and launder Modifications(A and B)

Rys. 10. Zbiornik z przegrodą i modyfikacjami płuczek (A i B)



(A)



(B)

Fig. 11 *Flow Pattern (A) and Sludge Blanket (B) (in a vertical section along tank central axis) in Tank with Modification 1, 2, 3, and 6*

Rys. 11. *Charakterystyka przepływu (A) oraz kożuch osadu (B) (w przekroju pionowym wzdłuż środkowej osi zbiornika) w zmodyfikowanym zbiorniku 1, 2, 3 i 6*

The predicted flow and solids fields in the tank with modification 1, 2, 3 and 6 are presented in Figures 11 (a) and (b). The flow pattern shows that influent density waterfall and surface reverse flow were significantly reduced by the 3 baffles. The flocculation baffle eliminates most of the entrainment flow from the surface clear water layers into the influent density flow thus, both the surface return flow along the entire tank surface and the bottom density current are substantially reduced [see Figures 9(A) and 11(A)].

The distribution of the sludge blanket among the 3 Bays has been significantly changed by using two perforated baffles [see Figures. 9 (B) and 11 (B)]. In the flocculation zone relatively minor solids compression takes place in the local sludge blanket. The highest sludge blanket occurs in Bay A due to the high resistance of the perforated baffle A/B. The lowest sludge blanket appears in Bay C. The difference of the sludge blanket level between Bay B and C is relatively small due to the lower resistance of perforated baffle B/C. The predicted flow pattern and solids field in Figures 11(A) and 11 (B) show that solids, spill gently over the baffle slots, at a lower velocity and potential energy head then that in the upstream Bays.

Tab. 2 Summary of Loading and Effluent Concentration in Tank

Tab. 2. Podsumowanie stężenia dopływowego i odpływowego w zbiorniku

	Q= 50 l/s Influent conc.= 50 mg/l	Q= 70 l/s Influent conc.= 50 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	28	34	50	50
Modification 1, 2, and 3	10	12	33	22
Modification 1, 4, and 5	7	9	25	14
Modification 1, 2, 3, and 6	5	6	11	9

6. Behavior of Tanks with Modifications in Holic ST

Because of the kinetic energy dissipation in the bottom density current due to the perforated baffles, greater solids compression occurs in Bay B and C. The solids distributions on the surface layer in the tank with modification 1, 2, 3, and 6 are presented in Figure 12. The simulated solids distributions indicate that the relocated effluent launders can avoid the sludge blowing out at the two downstream corners. Extending launder length more evenly distributes the effluent flow.

The model results in Table 2 show that the greatest effectiveness in reducing the effluent

solids using baffles alone was obtained when the tank is operated in stage 1 (the sludge blanket was lower than 40% of the water depth). For stage 2 operation since the effluent concentration was primarily controlled by the sludge blanket, the baffle's efficiency was dramatically reduced. For most cases the effluent concentration in the tanks with modifications 1, 2, 3 and 6 can be reduced to less than half of that in the current tank for the same loading. The results in the tanks with combinations to flocculation baffle and various inter-bay baffles are also included in Table 2. The performance of the tanks with the intermediate baffles was sensitive to the sludge blanket stage. Exceptionally good improvement in the tank performance was predicted for both solid (conventional) and perforated baffles for Stage 1 or low sludge blanket operation; for Stage 1, both perforated and solid baffles gave about 50% reduction in Effluent Suspended Solid (ESS). However, under high solids loading, i.e., Stage 2 or high blanket operation, the baffles became less effective and produced a short circuiting that in some cases degraded the effluent SS. The perforated baffle appeared to marginally better than conventional baffle in Stage 1 and significantly better in Stage 2 since the perforated baffles keep more sludge at upstream of the tank.

The all scenarios cases developed in the study not only provide the project owner a solution with the best effectiveness but also let them have a choice requiring a very limited cost while providing reasonable performance enhancement.

The relationship between the effluent concentration and the hydraulic loading is summarized in Table 2 for the existing tank and tanks with 6 different modification combinations. The predicted effluent concentration in Table 2 indicates that the average effluent concentration can be significantly reduced by improving the tank hydraulic efficiency.

Figure 13 indicates that the average effluent concentration can be significantly reduced by improving the tank hydraulic efficiency. The results demonstrate the effect of use the effluent launders on the effluent quality. The tank effluent with the highest solids concentration occurs at the downstream where the upward current, which is focused from both directions, is much stronger than elsewhere. This interesting phenomenon also was observed by the tank operator at Holic WTP and they describe is as "Sludge blows out".

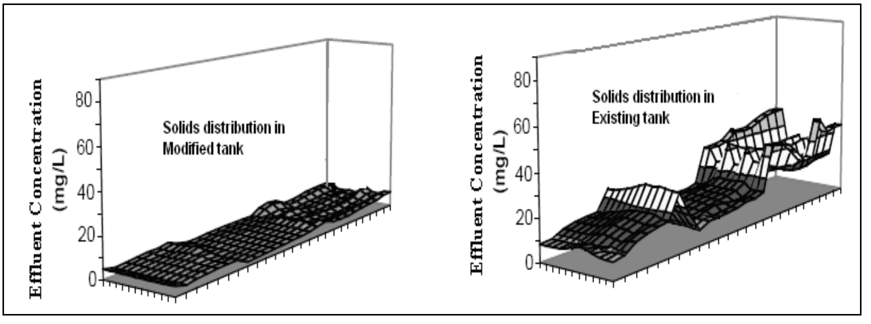


Fig. 13 Comparisons of Solids Distributions on Surface Layer between Existing and Modified Tanks (1, 2, 3, and 6)

Rys. 13. Porównanie rozkładu substancji stałych na warstwie wierzchniej między zbiornikami istniejącymi a zmodyfikowanymi (1, 2, 3i 6)

7. Conclusion

The 2-D fully mass conservative ST model is applied to predict the tank performance in the existing STs and the STs with proposed modifications at the Holic Water Treatment Plants. The existing Holic STs suffered from the relatively poor hydraulic performance which typically occurs in a large tank without proper baffling. The unfavorable hydraulic regime includes strong turbulence, a high influent potential energy and a strong density current due to excessive flow entrainment. The baffle modifications can considerably reduce the strength of the density flow and increase the solids detention time in the tank; the effluent quality can be improved by more than 60% for any cases. Proper launder modifications can be used to improve local flow pattern near the effluent weir and to re-distribute the effluent flow along the tank longitudinal direction.

Using the CFD modeling results together with the field work presented in this study, the following conclusions can be obtained as:

1. The use of an inboard launder also produced a significant benefit in terms of reduced ESS. The best result was obtained by a combination of inboard launder and a perforated baffle.

2. In general, CFD can be a powerful tool for troubleshooting problems, particularly those associated with flow patterns in a sedimentation tank. The results of this work give an insight, which can be used to investigate novel designs or different operating conditions, such as temperature variation, for production-scale tanks. Of course, because of the complexity of the processes taking place, CFD will not completely replace experimental testing and the partly empirical nature of the design process.

8. Acknowledgement

The Research Grants VEGA No. 1/3313/06 and projected KEGA 3/5125/07 solved at the Department of Sanitary and Environmental Engineering, Faculty of Civil Engineering, Slovak University of Technology, Bratislava have supported this project.

References

[1] Dahl, C. Larsen, T. & Petersen, O. Numerical modelling and measurement in a test secondary settling tank, Wat. Sci. Tech. 30(2), 219-228. 1994

[2] Ghawi A. Hadi & Jozef Kris. Design and Optimization of Sedimentation Tank in Slovakia with CFD Modeling. 10th International Symposium on Water Management and Hydraulic Engineering 2007 with special emphasis on the impact of hydraulic engineering construction on the environment 4 – 9 September 2007. Šibenik , Croatia. 2007a.

[3] Ghawi A. Hadi, & Jozef Kris. A Numerical Model of Flow in Sedimentation Tanks in Slovakia" Third International PhD Symposium in Engineering 25-26 October 2007, Hungary University of Pollack Mihály Faculty of Engineering Pécs, Hungary. 2007b.

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

[5] Larsen P. On the hydraulics of rectangular settling basins, Report No. 1001, Department of Water Research Engineering, Lund Institute of Technology, Lund, Sweden. 1977.