PERFORMANCE IMPROVEMENT OF WATER TREATMENT PLANTS IN IRAQ BY CFD MODEL

Dr.Abbas A. Al-Jeebory
College of Engineering
AL-Qadissiyia University

Dr. Josef Kris
College of Civil Engineering
Slovak University of Technology

Dr.Ali H. Ghawi
College of Engineering
AL-Qadissiyia University

Abstract

The aim of this study is to improve the operation and performance Water Treatment Plant by improving circular sedimentation tanks of Al-Gazaer Water Treatment Plants in Al-Dewanyia city in Iraq which have been identified as operating poorly. A developed model of the circular settling tanks for the Water Treatment Plant of Al-Gazaer was prepared using Computational Fluid Dynamics (CFD). A three dimensional, multi-phase simulations with solids transport and removal included is used, and reflected the state of the art in settling modeling. Computational Fluid Dynamics simulations are employed to assess the effect of adding a vertical baffle at the feed section of a full-scale sedimentation tank for the improvement of solids settling in potable water treatment**.** Special attention was paid to the inlet baffle in the model setup. Apparently, there is a good agreement between measured and predicted values. Results show, the overall solids removal efficiency increased when using the baffle from 50 to 90.5% leading to a reduction of the effluent solids concentration of approximately 86%.

Keywords

Circular settling tank, computational fluid dynamics (CFD), residence time distribution (RTD), modeling, simulation

تحسين أداء محطات معالجة مياه الشرب في العراق باستخدام نموذج ديناميكا السوائل الحسابية

د. عباس عليوي الجبوري
كلية الهندسة – جامعة القادسية

د.جوزيف كرش
الجامعة التكنولوجية سلوفاكيا
كلية الهندسة

د . علي هادي غاوي
كلية الهندسة – جامعة القادسية

الخلاصة

تهدف هذه الدراسة إلى تحسين أداء محطات معالجة مياه الشرب في العراق و ذلك بتحسين أداء أحواض الترسيب في محطة الجزائر لمعالجة مياه الشرب في مدينة الديوانية و التي تعمل بشكل سيئ أي أن كفاءة المعالجة قليلة. تمت نمذجة أحواض الترسيب عن طريق تطوير نموذج ديناميكا السوائل الحسابية و الخاص بأحواض الترسيب. ان النموذج الرياضي الذي طور هو نموذج ثلاثي الأبعاد, و تضمن العلاقة بين المواد الصلبة و السائلة, و حساب تركيز المواد الصلبة في الحوض و تضمن حساب كفاءة أحواض الترسيب. في هذه الدراسة تم دراسة تغير طول و شكل مصدات دخول المياه الى حوض الترسيب و مدى تأثيرها على تحسين كفاءة الأحواض باستخدام نموذج ديناميكا السوائل الحسابية. وتم مقارنة النموذج الرياضي مع القياسات الحقلية لغرض معرفة مدى دقة النموذج ووجد ان هناك تطابق كبير و مريح بين النموذج الرياضي و القياسات الحقلية. ومن النتائج التي تم الحصول عليها في هذه الدراسة ان كفاءة أحواض الترسيب في معالجة مياه الشرب قد ازداد من 50% الى 90.5% و ان تركيز المواد الصلبة الخارجة من حوض الترسيب قلت بنسبة 86%.

Nomenclatures

Symbol	Description
F	Volume force term (N/m^3) which is zero in both the x and y directions.
U	Average flow velocity vector (m/s)
P	Average pressure (Pa)
η	Dynamic viscosity (Pa·s)
ρ	Density (kg/m^3)
t	Time (s)
C_μ	Model constant
k	Turbulent kinetic energy (m^2/s^2)
ε	Dissipation of turbulent energy (m^2/s^3)
C	Concentration of solids (mg/l)
U_s	Settling velocity (m/s)
σ_c	Schmidt number (0.7)
ν_t	Turbulent viscosity
U_{so}	Reference settling velocity (m/s)
r_h , r_p	Induce the domination of the first and the second term for the falling and the rising part
C_{ns}	Nonsettleable concentration (mg/l)
C_D	Drag coefficient
ρ	Fluid density (kg/m^3)
V_t	Blade-to-fluid relative velocity (m/s)
A	Scraper displacement area (m^2)
G_b, G_k	Bouncy and kinetic energy effect
CFD	Computational Fluid Dynamics
RTD	Residence Time Distribution
SS	Suspended Solids
TS	Total Solids

Introduction

The capital infrastructure for water treatment will have to be increased considerably in the immediate future. It is important that new and existing plants are designed to operate as efficiently as possible. The increased activity in this sector will put pressure on experienced designers. One way to promote efficient design and to de-bottleneck existing equipment is to use modern computer techniques.

Design of sedimentation tanks for water 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), Ghawi and Kris (2007 A), and Ghawi and Kris (2007 B)** 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).**

Recently, Computational fluid dynamics (CFD) software has become easy to use, fast and user-friendly. This new generation software offers an inexpensive means of testing and optimizing hydraulic operation of both existing constructions and those under design.

The aim of this study was originally stated as to improve the operation and performance Al-Gazaer Water Treatment Plants by improving circular sedimentation tanks of Al-Gazaer Water Treatment Plants in Al-Dewanyia city in Iraq which have been identified as operating poorly, which is achieved by predicting the existing flow distribution settling velocity of the sedimentation tanks using Computational Fluid Dynamics (CFD) techniques. FLUENT 6.3 was used for the case study of the effect of adding a feed flow control baffle on the efficiency of solids removal.

Material and Method

A full-scale circular sedimentation tank was investigated, similar to that used in the potable water treatment plant of Al-Dewanyia city. The plant receives raw water from of Al-Dewanyia river and its capacity is around 2000 m^3/hr. The employed processes include, flash mixer, coagulation–flocculation, sedimentation (4 sedimentation tanks), rapid sand filtration, and chlorination (**Figure 1**). The sedimentation tank, with a volume of 1030 m^3, is centre-fed with a peripheral weir. The bottom floors have a steep slope of 12° and a blade scraper pushes the sludge towards a central conical sludge hopper. Two tank configurations have been considered, One with only a small vertical baffle to guide the feed of the tank henceforth referred to as standard tank (**Figure 2a**). And another where the small baffle is extended by an inclined and a second vertical section, altogether meant to guide the fluid significantly deeper inside the tank, henceforth referred to as modified tank (**Figure 2b**).

All the tanks had an inlet perforated baffle and an effluent v-notch weir. **Table 1** shows the settling tanks data. **Table 2** shows the physical and hydraulic data during study periods.

Influent and effluent samples were collected at different operating periods. The liquid temperature ranged between 23-29 °C during the experiment. The samples were analyzed according to procedures outlined in "Standard Methods For The Examination of Water" , 17th edition, APHA, (1989) to determine the following parameters: Suspended Solids (SS), and Total Solids (TS) (Floc Concentration)

Methodology

1. CFD Modelling

Several computer software programs have been developed for Computational Fluid Dynamic (CFD) modelling. FLUENT 6.3 and the 3D k-ε turbulence model in the Environmental Engineering Module was used. During this study hydraulic CFD modelling began with the definition of settling tank geometry. Fluid characteristics and boundary conditions were defined. The momentum balance including the turbulence model and continuity equations were then solved numerically for the tank using the finite volume method. Finally, the obtained solution was post-processed to be properly visualised. Common mathematical hydraulic model equations used for CFD modelling include the momentum balances for a non-compressible viscous media and the continuity equation (Wilcox, 1998).

$$r\frac{\partial U}{\partial t}-\nabla\left[\left(m+rC_m\frac{k^2}{e}\right).(\nabla U+(\nabla U)^T)\right]+rU.\nabla U+\nabla P=F \quad (1)$$

$$\nabla.U=0 \quad (2)$$

In the settling model an additional scalar equation was added to include the concentration of the solids. This convection-diffusion equation is as follows:

$$r\frac{\P C}{\P t}+\frac{\P r(U+U_S)C}{\P x_i}=r\frac{\P}{\P x_i}\left(\frac{v_t}{s_c}\frac{\P C}{\P x_i}\right) \quad (3)$$

The settling velocity was modeled using the exponential settling function of Takács 1991, this expression being introduced in the resolution of the concentration equation.

$$U_S = U_{S0} X \exp[-r_h(C - U_{ns})] - U_{S0} X \exp[-r_p(C - U_{ns})] \quad (4)$$

The standard k-ε eddy-viscosity model is used to account for turbulent effects. The turbulent viscosity is defined as function of the turbulent kinetic energy k and its dissipation rate ε by the equation (Wilcox, 1998):

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (5)$$

The distributions of k and ε were determined from the following transport equations:

$$\frac{\partial \rho k}{\partial t} + \frac{\partial k}{\partial x_i}(\rho k u_i) = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x_i} \right) + G_k + G_b - \rho\varepsilon - Y_M + S_k \quad (6)$$

$$\frac{\partial \rho \varepsilon}{\partial t} + \frac{\partial k}{\partial x_i}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left((\mu + \frac{\mu_t}{\sigma_\varepsilon}) \frac{\partial \varepsilon}{\partial x_i} \right) + C_k \frac{\varepsilon}{k}(G_k + C_{sc} G_b) - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k} + S_\varepsilon \quad (7)$$

The model constants (C_μ, $C_{\varepsilon 1}$, $C_{\varepsilon 2}$, σ_k, σ_ε) in the above equations have been determined from experimental data and are set to standard parameters (Wilcox, 1998):
$C_\mu = 0.09$, $C_{\varepsilon 1} = 0.1256$, $C_{\varepsilon 2} = 1.92$, $\sigma_k = 0.9$, $\sigma_\varepsilon = 1.3$
G_b describes the influence of buoyancy effects and is defined as a function of the suspended solids concentration gradient:

$$G_b = \beta g \frac{\nu_t}{\sigma_c} \frac{\partial C}{\partial x} = \frac{\rho_p - \rho_w}{\rho_p \rho_w} g \frac{\nu_t}{\sigma_c} \frac{\partial C}{\partial x} \quad (8)$$

The concentration gradient, which reaches maximum values at the interface between the clear fluid and the sludge blanket, hinders turbulence. The source term G_b introduced in turbulence equation addresses this matter. The value of $C_{\varepsilon 2}$, usually reported as constant, varies with the ratio of gravity direction parallel flow velocity with respect to perpendicular flow velocity (Wilcox, 1998):

$$C_{SC} = \tanh\left|\frac{v}{u}\right| \quad (9)$$

The later expression yields values close to unity for unstable areas, and tends towards zero for stratified sedimentation. A Boussinesq-type approach also implies that the effect of sludge gravity is introduced implicitly as a function of suspended solids concentration. Its implementation in the momentum equations is carried out by means of source terms:

$$g(\rho_p - \rho_w) = gC \frac{\rho_p - \rho_w}{\rho_p} \quad (10)$$

The dependence of viscosity on concentration is empirically inputted at different concentration ranges. The effect of the scraper blades has been usually either neglected or introduced as uniform constant sources, especially in the modeling of circular sedimentation tank. However, due to the significance of the scraper system for a circular sedimentation tank, an additional sub-model is incorporated to better model the effects of solids transport. The conveying force exerted on the fluid is approximated as a function of fluid velocity including a flow regime dependent drag coefficient:

$$F_D = C_D \frac{1}{2} rAV_r^2 \tag{11}$$

2. Full-Scale Tracer Test

Tracer tests were performed using pulse addition of lithium chloride (LiCl). Trace concentrations extracted from the outlet can be plotted against time; this is refer to as residence time distribution curve (RTD). A mass of 10.0 kg LiCl was dissolved in water and diluted to form a 25 liter brine. The brine was poured into a 50 m long hose. By using pressurized wash-water, the tracer was injected into the inlet of the first tank, all within a few seconds. Approximately 150 samples (100 ml each) were taken during the tracer tests. Samples were taken at the outlet of the settling tank. The samples were allowed to settle and the supernatant was filtered (1.2 μm membrane filter, Titan 2 HPLC Filter Orange 30 mm) in order to reduce interference of solids. The lithium concentrations of the samples were measured using a flame photometer (Eppendorf ELEX 3631). This was calibrated on site, using final effluent as dilutant when creating a lithium standard curve.

3. Simulation and Boundary Condition

To limit computational power requirements, the circular settling tank was modeled in 3D. The major assumption in the development of the model is that the flow field is the same for all angular 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. A grid dependency study was performed to eliminate errors due to the coarseness of the grid and also to determine the best compromise between simulation accuracy, numerical stability, convergence, and computational time. In addition, the mesh density was chosen such that the grid was finest where velocity gradients are expected to be largest. The selected grid was comprised of 117,324 quadrilateral elements. Two other grids (one finer with 200,850 elements and one coarser with 9160 elements) were also used to determine the effect of the overall grid resolution on predictions. While the predictions obtained using the coarse grid were found to be different from those resulting from the selected one, the difference between the predictions made by the selected and fine grids were insignificant. As a result, the solutions from the grid of 117,324 quadrilateral elements were considered to be grid independent.

The segregated solution algorithm was selected. The settling tang k–ε turbulence model is 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 (**Wilcox, 1998**). 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.

As a first step, the fluid mechanics problem was solved in the absence of particles to find the steady state flow field. The converged solution was defined as the solution for which the normalized residual for all variables was less than 10^6. In addition, the convergence was checked from the outflow rate calculated at each iteration of the run. The convergence was achieved when the flow rate calculated to exit the tank no longer changed. The inlet was specified as a plug flow of water at $0.075 m.s^{-1}$, whereas the inlet turbulence intensity was set at 4.5%. The outlet was specified as a constant pressure outlet with a turbulence intensity of 6.0%. The water flow rate was $0.25\ m^3\ s^{-1}$. Based on this rate, the inlet mass flow rate of particles was estimated as $0.11\ kg\ s^{-1}$ using a measured solids concentration of $200\ mg.\ L^{-1}$, whereas the primary particle density was 1062 $Kg.m^{-3}$.

Results and Discussion

As far as the CFD model validity is concerned, **Figure 3** presents a comparison between data experimental measurment and the simulated values of the Li distribution in the effluent of the standard tank. Apparently, there is a good agreement between measured and predicted values.
The removal efficiency in settling tanks depends on the physical characteristics of the suspended solids as well as on the flow field and the mixing regime in the tank. Therefore the determination of flow and mixing characteristics is essential for the prediction of the tank efficiency. **Figure 4** presents the predicted streamlines for the standard and the modified tank. The influent, after impinging on the standard flow control baffle at point **A**, is deflected downwards to the tank bottom. The flow splits at point **B** on the bottom of the tank, producing a recirculation eddy at **C**. Generally, the flow pattern is characterized by a large recirculation region spanning a large part of the tank from top to bottom. Three smaller recirculation regions are also found; two at the top of the tank near the entry and exit points of the liquid stream and one at the bottom right-hand side of the tank just above the cavity where the sludge gathers before leaving the tank. These regions have a substantial impact on the hydrodynamics and the efficiency of the sedimentation tank. The same behavior was observed by **Stamou (1991)** in his flow velocity predictions in a settling tank using a curvature-modified k–ε model. The above-mentioned observations are in agreement with findings of **Zhou and McCorquodale (1992),** who studied numerically the velocity and solids distribution in a clarifier. According to another numerical work (**Deininger et al, 1998**), in secondary clarifiers there is a circular current showing: (1) forward flow velocities in the zone close to the tank bottom, (2) backward flow velocities in the upper zone of the tank, (3) higher forward flow velocities in the inlet than in the rim region, (4) higher backward flow velocities in the inlet than in the outlet region, (5) vertical currents downwards to the tank bottom in the inlet region, and (6) vertical currents upwards to the water level in the outlet region. For the case of secondary clarifiers with high-suspended solids concentration, a density current exists due to a higher density of the incoming suspension. This current sinks toward the sludge blanket right after leaving the inlet structure and flows towards the tank rim. As a result, backward velocities are induced in the upper water zone following the continuity equation. A number of researchers have observed the solids-cascading phenomenon in the clarification of concentrated activated sludge in either theoretical simulations or experimental works. As it can be concluded comparing **Figure 4(a)**, the particles do not affect the flow field. This observation is attributed to the particle loading in our sedimentation tank and is similar to that made by **Kim et al. (2005),** who worked in a secondary clarifier with a neutral density influent flow. On the contrary, in the case of high inlet fluid density (high solids concentration) combined with a low fluid velocity, the horizontal inlet flow does not even reach the flow control baffle, but plunges down toward the tank bottom as a density waterfall due to the low Froude number.
In the modified tank, the flow split point **B** moves more to the right of the tank bottom compared to the position in the standard tank and the recirculation zone above the sludge corner is now very small. It appears that the extended baffle does not affect the flow pattern or the particle trajectories throughout the tank or at the exit. Neither does it affect the particle settling patterns on the bottom of the tank. The difference is mostly restricted at the entrance section and near the bottom rim of the tank, so that the upward flow in the downstream zone is only slightly different.
In general, the extended baffle appears to provide better influent mixing and isolation between the tank influent and effluent than that in the original tank design, thus significantly enhancing sedimentation. In addition, it allows a better utilization of the full tank depth than in the standard design that leads to better separation between the influent and effluent along the vertical direction. Studies by **Zhou and McCorquodale (1992)** revealed the importance of a baffle in dissipating the kinetic energy of the incoming flow and reducing short-circuiting and indicated that the location of the baffle has a pronounced effect on the nature of the flow. The percents presented in **Table 3**

result in an overall settling efficiency of 50 and 90.5% for the standard and the modified tank, respectively.
The overall solids removal efficiency increased when using the baffle from 50 to 90.5 % leading to a reduction of the effluent solids concentration of approximately 86%. The increase in the overall effectiveness seems small it corresponds to an estimated reduction in the solids exiting the tank of approximately 610 kg d^{-1} or to a reduction of about 86% of the solids that exit the tank. These values are greater than those reported by other researchers. **Huggins et al. (2005),** who tested a number of potential raceway design modifications noticed that by adding a baffle the overall percent solids removal efficiency increased from 81.8 to 91.1% resulting in a reduction of the effluent solids of approximately 51%. **Crosby (1984)** used an additional baffle at mid-radius extending from the floor upwards to mid-depth and observed a reduction of 38% in effluent concentration. According to **Huggins et al. (2005),** the particle settling velocity has a significant impact on the settling efficiency for a given raceway design. These authors argued that an important consideration in trying to improve the settling of particles is to reduce the mass fraction of solids with settling velocities below $0.01 m.s^{-1}$.However, since influent solids load is usually uncontrollable one should focus instead on the design of a proper baffle, which will improve solids settling by forcing them to reach fast the bottom of the tank. **Figure 5** shows contours of velocity for the standard and the modified tank.
The effect of modifications are also displayed in**Figures 6** and **7** that show flocs concentration along the tank bottom. In **Figures 6** and **7** the zero position of the horizontal axis is set at the right-hand end of the tank bottom. Clearly, the modified tank allows flocs to settle at much short distances from the right-hand corner of the tank. This diminishes the overall settling efficiency of the tank. On the whole, the simulation results demonstrate quantitatively the drastic effect of particle velocity on sedimentation effectiveness. Higher settling velocities lead to more effective sedimentation. However, even small differences in particle settling velocity can cause large changes in the percent of settled particles

Conclusions

CFD could be used in reviewing settling tank design or performance and the results give valuable insight into how the tanks are working. Also CFD could be use to evaluate settling tank designs where the tanks are not functioning properly.
Overall, the following conclusions can be taken in relation to this study:

- CFD modeling is successfully used to evaluate the performance of a settling tank and water treatment plant.
- High solid removal efficiency were achieved.
- The results show that an extended baffle forces the solids to move faster towards the bottom of the tank and decreases the inlet recirculation zone, thus yielding significantly enhanced sedimentation. Although the increase in the overall effectiveness by this baffle may show only a small change, this actually reflects a reduction of the effluent solids of estimated around 86%.

Acknowledgements

The authors wish to thank the Al-Gazaer Water Treatment Plant staff for accessing to the required data.

References

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

Adams E.W., & Rodi W., Modelling flow and mixing in sedimentation tanks, J. Hydr. Eng. 116 (1990) 895–913.

APHA, "Standard Methods for the Examination of Water and Wastewater," 17th edition, American Public Health Association, Washington, D.C. (1989).

Crosby R.M., Hydraulic characteristics of activated sludge-secondary clarifiers, EPA 600/2-84-131, NTIS No. PB-84-229665, U.S. EPA, Municipal and Environmental Research Laboratories, Cincinnatti, OH, (1984).

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 (2005) 167–180.

Deininger A., Holthausen E., & Wilderer P.A. Velocity and solids distribution in circular secondary clarifiers: full scale measurements and numerical modelling, Water Res. 32 (1998) 2951–2958.

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

Ghawi A. Hadi & Jozef Kris. Design and Optimization of Sedimentation Tank in Slovakia with CFD Modelling. 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).

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).

Kim H.S., Shin M.S., Jang D.S., Jung S.H., & J.H. Jin, Study of flow characteristics in a secondary clarifier by numerical simulation and radioisotope tracer technique, Appl. Radiat. Isotopes 63 (2005) 519–526.

Launder B.E., & Spalding D.B., The numerical computation of turbulent flows, Comp. Meth. Appl. Mech. Eng. 3 (1974) 269–289.

Stamou A.I., On the prediction of flow and mixing in settling tanks using a curvature modified k-ε model, Appl. Math. Modelling 15 (1991) 351-358.

Takács I., Patry G.G. & Nolasco D. (1991). A dynamic model of the clarification-thickening process. Wat. Res., 25(10), pp. 1263-1271.

Wilcox D.C., Turbulence Modeling for CFD, DCW Industries, Inc., La Canada, California, (1998).

Zhou S., & McCorquodale J.A., Influence of skirt radius on performance of circular clarifiers with density stratification, Int. J. Numer. Meth. Fluids 14 (1992) 9-19.

Table 1. Settling tanks data.

Tank parameter	Value
Tank diameter (m)	18 m
Depth of inlet (m)	5.6 m
Tank depth (m)	4.0 m
No. of tanks	4

Table 2. Physical and hydraulic data during study periods.

Tank parameter	Value
Average flow rate	500 m^3/hr
Sludge pumping rate	5 l/s
Average Inflow temperature	6^oC in winter and 29^oC in summer
Inflow suspended solids	25-75 mg/l
Detention time	3.5 hr
C_{min}	0.17 mg/l

Table 3. Performance data for modelled settling tank

Efficiency Of Settling Tanks	Influent Conc.= 40 mg/l	Influent Conc.= 50 mg/l	Influent Conc.= 60 mg/l	Influent Conc.= 75 mg/l
	Predicted average effluent concentration			
Existing Tank (50 %) (from measurement)	20	25	30	37.5
Modified Tank (90.5 %) (from CFD)	3.8	4.7	5.7	7.1

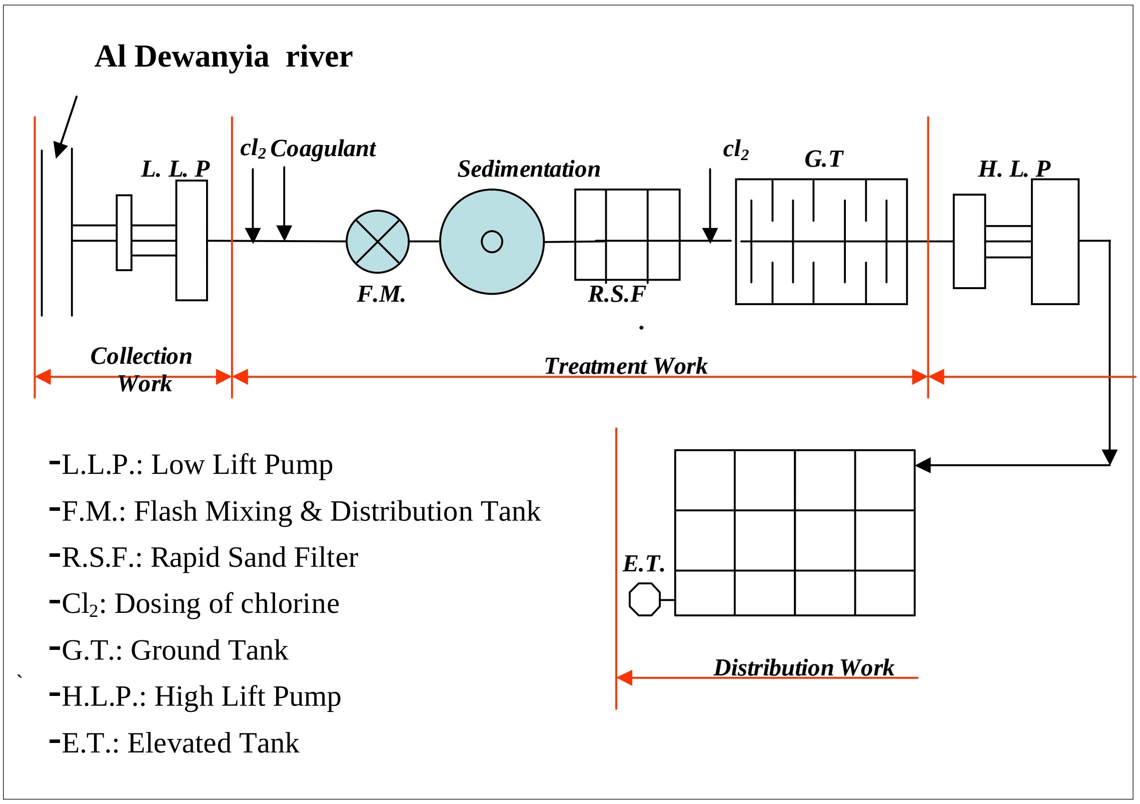


Figure 1. Layout of water treatment plant of Al-Gazaer in Al-Dewanyia city.

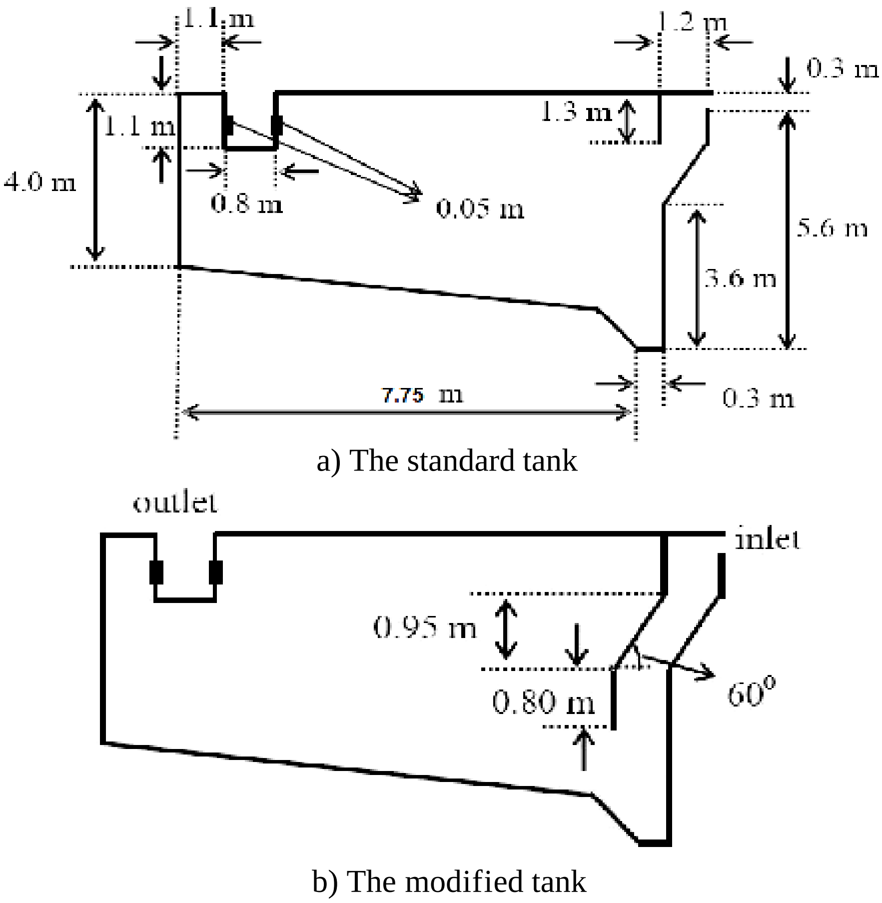


Figure 2. Schematic representation of the standard (a) and the modified (b) simulated sedimentation tank.

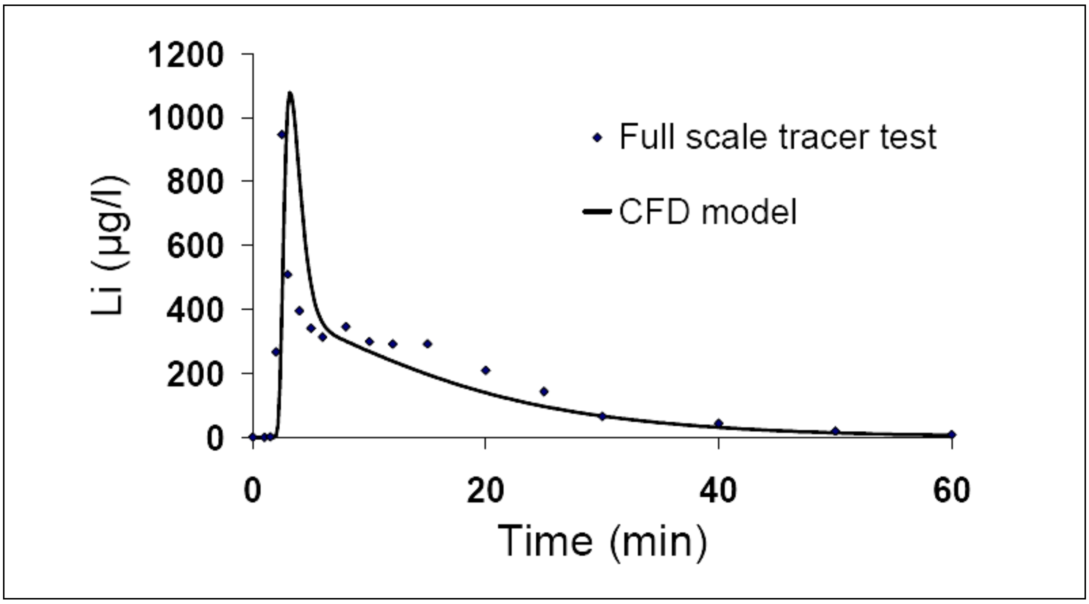
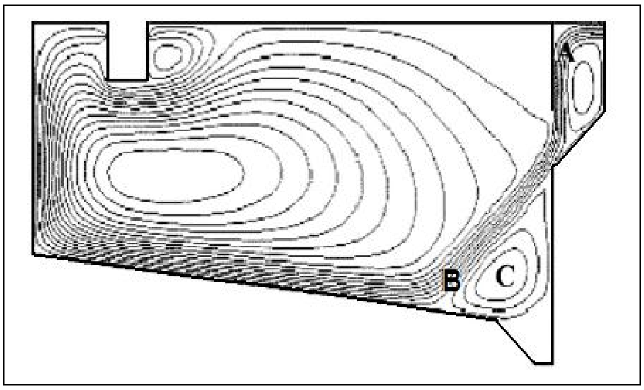
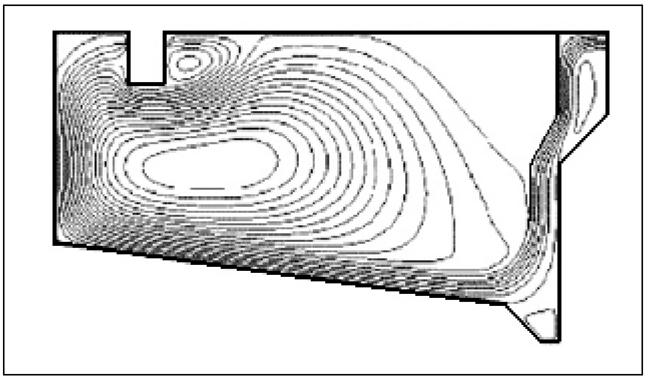


Figure 3. Li tracer characteristic of the standard sedimentation tank (concentration–time relationship).



a) Standard tank



b) Modified tank.

Figure 4. evaluated streamlines for sedimentation tank from CFD result. (a) Standard tank, and (b) Modified tank.

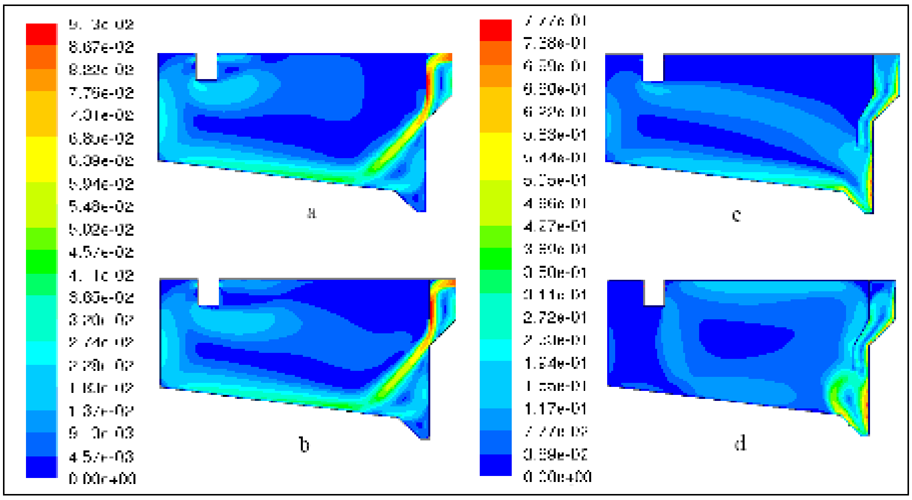
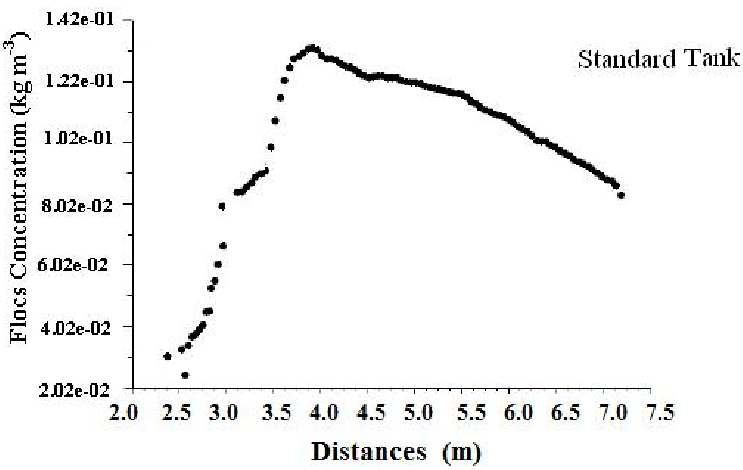
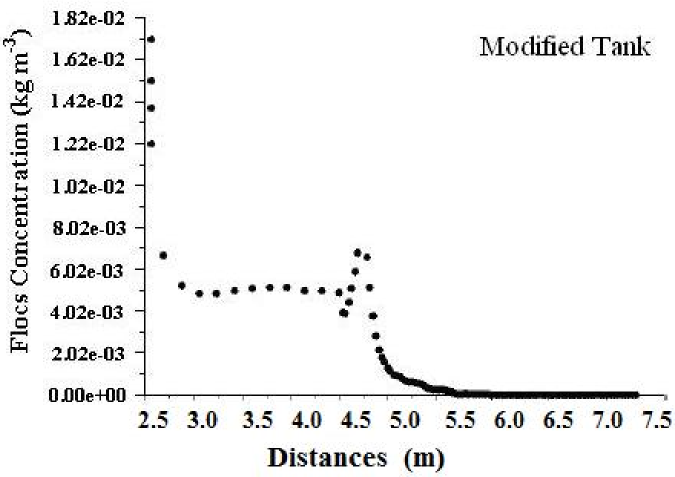


Figure 5. velocity Contours (m s^{-1}) for sedimentation tank for (a,b) the standard and (c,d) the modified tank for different solid concentration from (25 (a,b)-75 (c,d) mg/l)

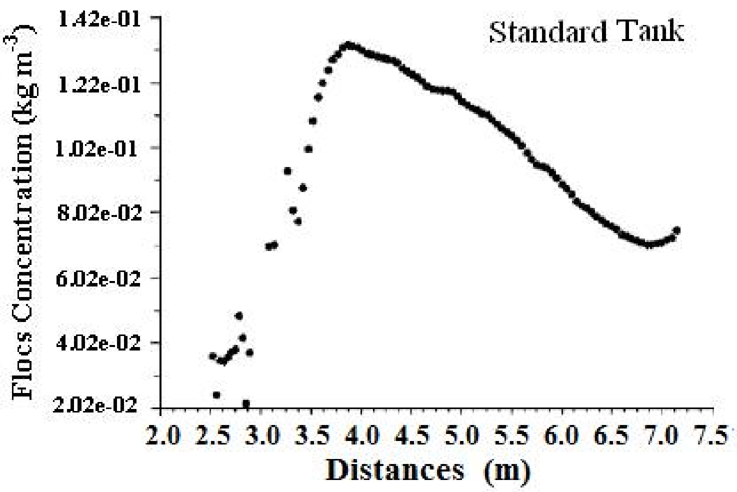


a)

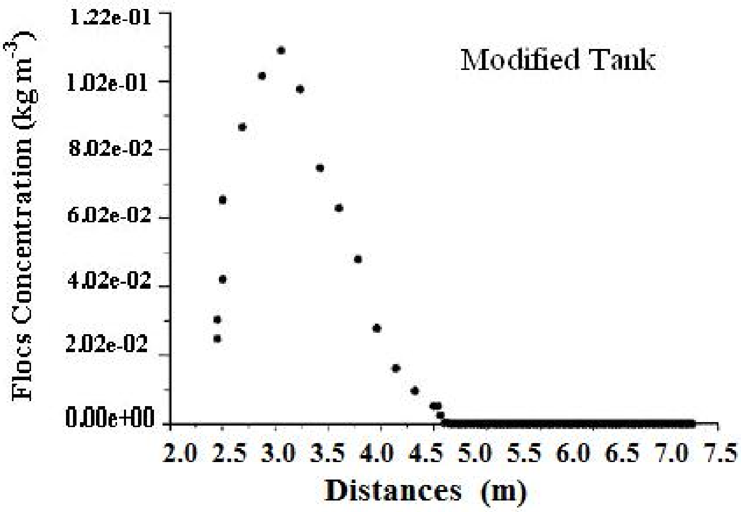


b)

Figure 6. Flocs concentration (kg m^{-3}) along the tank bottom for the standard (a) and the modified tank (b) for inlet concentration 40 mg/l.



a)



b)

Figure 7. Flocs concentration (kg. m^{-3}) along the tank bottom for the Standard (a) and the Modified Tank (b) for inlet concentration 75 mg/l