A COMPUTATION FLUID DYNAMIC INVESTIGATION INTO THE USE OF BAFFLES IN POTABLE WATER TREATMENT PLANT PROCESS TANKS IN IRAQ

ALI H. GHAWI PhD
College of Engineering
AL-Qadisiya University

Abstract

The reaction that takes place in a process tank is in a large way dependent on the internal hydraulics of the tank, irrespective of the type of tank. In some tanks mixing is desirable in others plug flow is desirable. In most tanks it is very hard to achieve either of the two. This is partially due to the fact that the area through which the fluid enters the tank is small in relation to the tank cross sectional, complex geometries and changes of fluid direction. Baffles can in some cases offer cost effective and simple solutions. This paper demonstrates how Computation Fluid Dynamic (CFD) can be used to improve process tanks performance in Iraq water treatment plant by the introduction of baffles.

Keyword: **baffle, sedimentation tanks, reservoirs, chlorine contact tanks, flocculates**,

تحقيق ديناميكية حسابِ المائع في استعمالِ الحواجز في أحواضِ محطات معالجةِ الماءِ الصالح للشرب في العراق

د. علي هادي غاوي
كلية الهندسة
جامعة القادسية

الخلاصة

التفاعل الذي يَحْدثُ في أحواض معالجة مياه الشرب عَلى نَحوٍ كبير معتمدة على الهيدروليكية الداخلية للأحواض، بصرف النّظر عن نوعِ الحوض. في بَعْض الأحواض تكون عملية الجريان الممزوج مرغوبة و البعض الأخر تكون عملية الجريان المتدفقِ مرغوبُة. في أكثر الأحواض من الصعب جداً ان يحصل الاثنان معا". هذهً الحقيقة بسبب إن منطقة دخول المائع إلى الحوض صغيرة بالنسبة إلى مساحة الحوض, الشكل المعقدة للحوض و التغير في اتجاه الجريان . المصدات لها القدرة على تقلل كلفة معالجة الماء الصالح للشرب في محطات المعالجة و كذلك توفر حلول سهلة و بسيطة في عملية المعالجة و لمختلف أحواض المعالجةِ. هذه الدراسة أثبتت كيفية ان ديناميكية حساب المائع (CFD) يمكن أن تُستَعملَ لتحسّينُ أداءُ الأحواض المستخدمة في محطات معالجة مياه الشرب في العراق و بتالي الحصول على نوعية مطابقة للمواصفات العالمية لنوعية الماء الصالح للشرب باستخدام المصدات.

Nomenclature

ρ_r	reference density (clean water).
u_0	influent velocity
H_{in}	depth of influent stream
C	solid concentration
S_S	specific gravity of the solid particles
k	turbulence kinetic energy
ε	dissipation of turbulence energy
φ	denotes general dependent variables expressed as a physical quantity per unit mass.
u, v	stand for x, y velocity components
ρ	density
$\Gamma\varphi$	diffusion coefficient
$S\varphi$	source term corresponding to φ
V_s	settling velocity
V_o	Stokes velocity (settling velocity of single particle in clear water)
K	an empirical coefficient for rapidly settling flocks
C_{min}	concentration of non-settling flocks
K_1	a settling exponent for the poorly settling particles
P	production of turbulent energy by the mean velocity gradients
C_1, C_2 , $C\mu$	k-ε model constants
σ_k , σ_ε:	turbulent Prandtl numbers for k and ε,
Q	constant flowrate,
T	theoretical mean residence time
C_i	tracer concentration
E(t)	normalized residence – time distribution function
R	normalised concentration of the tracer
F	densimetric Froude number
CFD	Computational Fluid Dynamic
RTD	Residence time distribution
SS	Suspended Solid

Introduction

Successful design and reliable operation of new water treatment facilities and the constant upgrading of existing plants has always been, and continues to be a major concern for all world water professionals. The main objective is to ensure the adequate treatment process and to guarantee its maximum efficiency.

In flow dynamics a baffle is a device used to alter the flow pattern in a tank. Baffle also means to be puzzled or perplexed - the state of mind designers are in when the process tank does not perform the way it was supposed to. Fortunately modifications can be made to the tank after construction to improve its performance. One of the most cost effective ways to achieve this is by means of a baffle. Unfortunately the selection of a baffle to suite a specific process tank is not a trivial exercise, especially with only the conventional design methods at hand. The degrees of freedom in terms of size, vertical and horizontal position and type of baffle make it almost impossible to make the right choice the first time. The effect of a baffles can normally only be seen after the baffle has been installed. Scale models can give some guidance, but are often plagued by scale effects which mean that what happens in the lab does not happen in the plant.

Computation Fluid Dynamic (CFD) modelling is being increasingly used in a wide range of both water and wastewater treatment applications. The CFD modelling work reported here was carried out using FLUENT software.

Very few guidelines exist for the optimal use of baffles in process tanks in WTPs. This shortcoming often results in baffles performing less effective for the following reasons:

- Conventional wisdom is often proved wrong due to a lack of understanding of the complex flow patterns in most process tanks.
- Empirical data is not available to give quantitative guidelines.
- Experimental trial-and-error type investigations are time consuming and costly.

What can be done to improve the situation? Before this question is addressed, a short digression on the use of baffles.

The aim of this project was originally stated as to improve the operation and performance of Iraqi Water Treatment Plant (WTP) which have been identified as operating poorly, by predicting the existing flow, flocculent and chlorine concentration distribution of the (Sedimentation, Reservoirs, Chlorine contact, and Flocculators tanks) by means of CFD techniques. Details about the model characteristics , initial and boundary conditions are presented in **Kris and Ghawi 2007**, **Ghawi and Kris 2007 a**, and **Ghawi and Kris 2007b**.

For this study the flow field is considered to be isothermal, incompressible and without phase change. The CFD package FLUENT 6.3.26 was used for the case study of the effect of adding a control baffle on the efficiency of tanks and then on efficiency of WTP in Iraq.

2. Material And Methods

1 How Baffles Are Used

The purpose of introducing baffles varies from tank to tank, but can include the following:

- In some cases it can be used to force the flow in such a way that the fluid moves more or less at the same velocity everywhere in the tank. Examples of such tanks would include tanks where contact between water and a disinfectant is required such as chlorine.
- Some process tanks require deliberate velocity gradients to enable collisions between suspended particles as in the case of a hydraulic flocculator. Baffles can be used to cause the flow to zig-zag or to spiral (**Figure 1a**).
- In other cases baffles are introduced to generate turbulence and improve the mixing of a chemical with the bulk fluid such as a static mixer (**Figure 1b**).
- Most of the above cases use solid baffles to direct the flow, but baffles can also be used to damp the inlet jets, by dispersing it over a larger area.

Perforated baffles can be used for this purpose as in the case of a sedimentation tank (**Figure 1c**).

Two types of baffles can therefore be distinguished:

- Solid baffles, used as obstructions perpendicular to the flow to change the direction of the flow.
- Perforated baffles, used to break up the main jet into a more uniform current over a larger area without the deliberate intention to change the direction.

In most cases baffles are designed as part of the process tank, but in some cases the reality check requires the introduction of additional baffles or modified baffles after the construction of the tank. Baffles are then used to deliberately separate the inlet and outlet to prevent short circuiting or improve the level of plug flow or mixing. Baffles can play an important part in the improvement of process tanks, and design tools to determine baffle placement can assist designers. Not only can it potentially save on the initial capital cost of a process tank, but it can also reduce the operating and maintenance costs. CFD provides a generic framework by which baffles can be designed. In this study we use different kind of baffles in different position in sedimentation tanks, reservoirs, chlorine contact tanks, and flocculators tanks in WTPs in Iraq to improve the operation and performance of WTPs. CFD model and Residence time distribution (RTD) model are used to analysis the flow and concentration of suspended solid, flocculent and chlorine in WTPs.

2 The CFD Model

The flow patterns for density-stratified fluids in the tanks are usually quite different from those with uniform density under the same external and boundary conditions. The relative importance of inertial and gravity forces in the tanks can be characterized in terms of the initial momentum and buoyancy flux defined by the inlet densimetric Froude number, F.

$$F = \left[u_0^2 / \frac{gH_{in}(\rho - \rho_r)}{\rho_r} \right]^{1/2} \quad \textbf{(1)}$$

in which ρ_r = the reference density (clean water). The value of ρ is the local density of mixture; u_0 = influent velocity; and H_{in}= depth of influent stream.
The local fluid density is related to the local values of sediment concentration given by

$$\rho = \rho_r + C(1 - S_S^{-1})$$

where C = solid concentration and S_S = the specific gravity of the solid particles.
A numerical model consists of a set of conservation equations for continuity and momentum, turbulence kinetic energy k, dissipation of turbulence energy ε and solid concentration. They can be expressed as in a governing equation

Continuity equation

$$\frac{\partial(\rho\phi)}{\partial t} + \frac{\partial}{\partial x_j}(\overrightarrow{\rho u_j \phi}) = \frac{\partial}{\partial x_j}(\Gamma_\Phi \frac{\partial \phi}{\partial x_j}) + S_\Phi \quad \textbf{(2)}$$

Suspended Solid (SS) concentration equation

$$\frac{\partial(C_i)}{\partial t} + u\frac{\partial C_i}{\partial x} + v\frac{\partial C_i}{\partial y} = \frac{\partial}{\partial x}(v_{zx}\frac{\partial C_i}{\partial x}) + \frac{\partial}{\partial y}(v_{zy}\frac{\partial C_i}{\partial y} + V_s C_i) \quad \textbf{(3)}$$

where φ denotes general dependent variables expressed as a physical quantity per unit mass. Further, $u, v, \rho, \Gamma\varphi$ and $S\varphi$ stand for x, y velocity components, density, diffusion coefficient and source term corresponding to φ, respectively. Eq (2) is a convection and diffusion equation for the suspended solid particles where V_s is the settling velocity.
The main feature of the density–induced flow field in settling tank is known to be strongly affected by solid distribution and removal rate. The settling velocity of a particle contained in a disperse suspension, as specially found in settling tank, is also dependent on both its individual characteristics (size, density, shape, etc.) and interactions with other particles. In low concentration, the effect of particle interaction is not significant and thus each particle settles as a separate entity while in high concentration, inter-particle force hinders the settling process of the particle, which settles as a unit, with decreasing mass velocity as the concentration increases.
Therefore modeling the settling properties of SS is one of the significant processes in the development of solid transport models of settling tank.

A number of empirical formulas have been proposed to describe the relationship between solid concentration and solid settling velocity. A monodisperse settling model in which the average settling velocity V_s of the settleable fraction of SS is given by the sum of two exponential terms:

$$V_s = V_0 \left[e^{-K(C-C_{\min})} - e^{-K_I(C-C_{\min})} \right] \qquad \textbf{(4)}$$

where V_o = Stokes velocity (settling velocity of single particle in clear water)
K = an empirical coefficient for rapidly settling flocks
C_{min} = the concentration of non-settling flocks
K_I = a settling exponent for the poorly settling particles
The first term in Eq. (4) reflects the settling velocity of the large particles, whereas the second term is a velocity correction factor to account for the smaller, slowly settling particles.

For low concentration, Eq. (4) is more sensitive to k_I and it gives increasing values of V_s as the concentration increases, whereas for high concentration it reduces to Vesilind's equation. The regions of low and high solid concentrations are separated by the point where Vs reaches a maximum value.

The eddy viscosity, v_t is calculated from the k-ε turbulence model which relates v_t to the turbulence kinetic energy of k and the turbulence dissipation rate ε. The formula is:

$$v_t = C_\mu \frac{k^2}{\varepsilon} \qquad \textbf{(5)}$$

The distribution of k and ε is calculated from the following semi-empirical transport equations.

$$\frac{\partial k}{\partial t} + u\frac{\partial k}{\partial x} + v\frac{\partial k}{\partial y} = \frac{\partial}{\partial x}\left(\frac{v_t}{\sigma_k}\frac{\partial k}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{v_t}{\sigma_k}\frac{\partial k}{\partial y}\right) + P - \varepsilon \qquad \textbf{(6)}$$

$$\frac{\partial \varepsilon}{\partial t} + u\frac{\partial \varepsilon}{\partial x} + v\frac{\partial \varepsilon}{\partial y} = \frac{\partial}{\partial x}\left(\frac{v_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x}\right) + \frac{\partial}{\partial y}\left(\frac{v_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial y}\right) + C_1\frac{\varepsilon}{k}P - C_2\frac{\varepsilon^2}{k} \qquad \textbf{(7)}$$

in which P is the production of turbulent energy by the mean velocity gradients as the k-ε model constants, C_1, C_2 and $C\mu$ as well as turbulent Prandtl numbers for k and ε, σ_k and σ_ε are selected as the conventional standard values of 1.42, 1.68, 0.09, 0.9 and 0.9, respectively.

Boundary conditions for u, v, k, ε and C_i are minutely presented in a previous work (**Kris and Ghawi 2007**) and are briefly mentioned for completeness. A uniform parallel inlet flow is assumed with specified values of k and

$$\varepsilon = C_\mu^{3/4}(k^{3/2}/l_m) \qquad \textbf{(8)}$$

The inlet SS loading is assumed to be in a well mixed state with dimensionless concentration unit. The outlet boundary values are computed from near outlet grid points by satisfying the overall mass continuity. Near wall velocities parallel to the wall are calculated using the local logarithm law.

Near wall k and ε can be calculated from the assumption of local equilibrium condition. The tank bottom is treated as a perfect absorbing boundary where particles may not be resuspended by the fluid flow.

3 Residences-Time Distribution (RTD) Model

Residence time distribution (RTD) analysis provides means of assessing the homogeneity of water age in a tank. A plot of the fraction of water leaving the tank outlet as a function of time can be used to assess the extent of short-circuiting and the existence of low flow zones in the tank. Traditionally, this has been measured by conducting a physical tracer study on the tank.
The following background equations and definitions are used in the proposed model.

Considering a volume V and constant flowrate Q, the theoretical mean residence time T is given by

$$T = \frac{V}{Q} \tag{9}$$

The normalized residence – time distribution function E(t) after an instantaneous (pluse) tracer input is defined as :

$$E = \frac{C_i}{\int_0^\infty C dt} \approx \frac{C_i}{\sum_{i=1}^{n} C_i \Delta t_i} \tag{10}$$

Where t denotes time, and C_i is the tracer concentration measured at the outlet at time t_i.An alternative to the pulse tracer study is the step tracer study. In this case, at t = 0 the tracer is "switched on" at the tank inlet and remains "on" for the remainder of the study. The normalised concentration of the tracer measured at the tank outlet under steady-state operation is R (Eq. 11). The relationship between E and R is given by Eq. 12.

$$R = \frac{C_i}{\int_0^\infty C dt} \approx \frac{C_i}{\sum_{i=1}^{n} C_i \Delta t_i} \tag{11}$$

$$R = \int_0^\infty E dt \tag{12}$$

While either a pulse or step physical tracer study can be used to determine the RTD of an operational tank, a physical tracer study can only be undertaken for operating conditions which are feasible in the physical tank. Other scenarios, such as the performance of the tank for operating conditions outside normal operating conditions, or the impact on the flow of physical tank modifications, cannot be assessed in advance using this method.

An equation was solved for "residence time", this being the mean age in the water in each cell since its entry to the tank. Contour plots of this variable give a good indication of how well mixed the water is, and whether there are any stagnant zones.

Result And Discussion

3.1 Baffle Dynamics

3.1.1 The Importance of Inlets and Outlets

One of the reasons why process tanks do not always behave according to designers' intuition, is because of the mechanisms at play near the inlets and outlets of process tanks. The inlet and outlet of a tank determines to a large extent the internal flow patterns in the tank. It is not often realized, but mechanisms applicable to inlets and outlets differ fundamentally.

- ***Inlets*** inject momentum (kinetic energy) into a process tank. This changes the mass inside the tank as well as the velocity of the surrounding fluid. The momentum is a vector and can therefore vary in terms of intensity as well as direction. The way in which the fluid enters a process tank often determines to a large extent the way in which the complete tank behaves. In some cases where more than one phase is present, as in the case of a sedimentation tank, not only kinetic energy is added to inlet, but also potential energy. This is caused by the density differences between the fluid and the suspended particles.
- An ***outlet*** on the other hand removes mass from the tank, but does not contribute to the removal of momentum in a similar way as at the inlet. Also the effect of an outlet is only local and does not normally affect internal hydraulics in the same way as the inlet. An outlet withdraws the fluid closest to it at the time.
- The fact that the inlet injects momentum and the outlet not, does not make an outlet less important. It is crucial to carefully select the position of the ***inlet in relation to the outlet*** to ensure that the process tank performs effectively. In many cases the process tank outlet is dictated by the external piping configuration and not the internal processes. This can have adverse effects on the internal flow patterns as well as the process efficiency.

This following section demonstrates how CFD can be used to improve process tank efficiency by introducing baffles. A number of process tanks that are typically used in the water industry are used as case studies to demonstrate this. The examples include:

- Sedimentation tanks
- Reservoirs
- Chlorine contact tanks, and
- Flocculators

The focus is on the insight gained by applying CFD and how it can be used to improve process tank efficiency for these tanks in some WTPs in Iraq.

3.2 Chlorine Contact Tanks and Clean Water Reservoirs

The purpose of a chlorine contact tank is usually two fold, the one is to act as a operational balancing storage device and the other is to achieve proper disinfection by forcing contact between the disinfectant and the water. Clean water reservoirs were traditionally considered to have an emergency and balancing storage function only, but they also act as large reactors in the distribution system. A number of studies demonstrated how poor internal hydraulics can influence water quality in a distribution reservoir. It was also demonstrated how the inlet and outlet configuration influences the internal hydraulics which in turn influences the water quality in the reservoir. The design objective is therefore to achieve a uniform as possible flow pattern to ensure that the fluid moves continuously and that sufficient contact between the water and the disinfectant is achieved. This can only be achieved if the internal hydraulics of the tank is properly understood. Two problems are usually associated with these types of tanks viz. short-circuiting of the inlet and outlet and the presence of stagnant areas. Most effective ways these tanks can be improved is by:

- Repositioning the inlet
- Repositioning the outlet
- Introducing internal baffles to force the flow in a certain direction

The CFD analysis of this distribution storage reservoir are done in accordance with a previous study (**Ghawi and Kris 2008 a**). The inlet and outlet arrangement was not changed, but three concentric baffles were added to reduce the extent of the stagnant areas. **Figure** 2 and 3 shows the reduction of total stagnant area.

3.2 Hydraulic Flocculators

The primary purpose of a hydraulic flocculator is to generate velocity gradients to stimulate inter particle collisions. Hydraulic flocculators are principally characterized by their volume (which determines the time of flocculation) and the water level difference between inlet and outlet (which determines the average velocity gradient). The underlying assumption is that the velocity gradient is proportional to the head loss. The local detail inside the flocculator is however disregarded in the process as only the global volume and global pressure drop is considered. The velocity gradients which are the primary force driving the flocculation process cannot be optimized with conventional methods as the designer has too many degrees of freedom, such as the average water depth, the number and spacing of baffles, the length of the gap at the baffle ends, and the degree to which adjoining baffles overlap (**Ghawi and Kris 2008 b**).

CFD allows the designer to evaluate the effect of changing the geometric ratios in an attempt to optimize the baffle placement. The channel length for instance can be stretched. Or the gap at the end of a baffle can be reduced. This enables the designer to calculate the optimal dimensions of a flocculator for a specific flow rate as described elsewhere. A typical flow pattern is shown in **Figure** 4.

3.3 Sedimentation Tanks

A sedimentation tank is a phase separating device that performs the following functions:

- Firstly it needs to leave a clarified layer of water near the surface in the vicinity of the overflow launders.
- Secondly it needs to settle sludge in the vicinity of the sludge hopper to ensure effective hydraulic removal. In some tanks mechanical removal is done in which case this requirement is less important.

The following challenges are faced to achieve an effective design:

- The presence of suspended matter at the inlet of a sedimentation tank causes density currents and complicates the flow patterns of a sedimentation tank.
- Special precautions are required to manage the density currents that occur as a result of the suspended solids. This is particularly prominent in the case of horizontal settling tanks where only a small portion of the fluid in the tank will move from the inlet towards the outlet at a uniform velocity.
- Particular attention should be given to the inlet of the sedimentation tank. Baffles of various forms are used to damp the flow at the inlet to restrict the density current. Fluid is also introduced as low as possible to limit density currents.
- Particular attention should be given to the positioning and shaping of hoppers.
- Particular attention should be given to the positioning and shaping of overflow launders.

Figure 5 illustrate the model simulated velocities and solids distributions for the modifications tanks. The modelled Effluent Suspended Solids (ESS) is summarized in Table 1.

These simulations demonstrate that the inboard placement of the launder in a rectangular ST is a major factor affecting the tank performance (**Figure 5**). The role of the inlet baffle is important in two aspects: 1) This baffle reduces the re-entrainment of already settled liquid into the inlet zone, and 2) It provides a zone for flocculation. The perforated baffle interrupts the bottom density current and helps to redistribute the flow over the tank depth.

However this type of baffle may create its own bottom current which then partially negates its positive effect. For these trials the best combination was an inboard launder with a perforated baffle. Numerous studies have been conducted (e.g. **Ghawi and Kris 2008c**) to investigate the optimal placement of inlet baffles (reaction baffles) in secondary settling tanks.

3.4 Potable Water Storage Reservoirs

Water distribution systems include reservoirs which provide security of supply and even out supply and demand. Quality of the water supply is a prime requirement. Water quality can be adversely affected by a poor flow pattern in a reservoir, for instance if there are any significant stagnant zones or short-circuiting pathways. CFD has the potential to provide valuable design insight into the hydraulics of reservoirs, which is difficult to acquire from experimental techniques.

The flow pattern for the case without baffles is shown by means of velocity vectors in **Figure 6**, which correspond to flow at the floor. After impinging on the reservoir floor, the flow fans out radially. It moves outwards and upwards to the perimeter of the reservoir, is guided around by the perimeter, and then turns back along the main diameter towards the inlet. A significant feature of the flow pattern at the floor is a separation line where the incoming flow lifts off the floor, when it meets water returning in the opposite direction.

Figure 7 shows residence time of water at the surface. The residence time shows the water is well mixed.

For the case with baffles, velocity vectors and residence time for the water surface are shown in Figures 8 and 9. There are two parallel baffles, oriented at approximately 40^{o} to the diameter joining inlet and outlet, and arranged so as to form an S-shaped path for the water.

The velocity vectors show that the momentum of the inlet drives a relatively energetic recirculatory flow upstream of the first baffle. Outward flow past this baffle lies close to the perimeter wall, and continues along it to the second baffle. Between the baffles, and downstream of the second baffle, lie recirculation regions of relatively quiescent water.

This maximum residence time is much greater than for the case without baffles. The reason for this is that the baffles tend to suppress the mixing driven by the inlet momentum. The energetic flow near the inlet is largely screened from the remainder of the reservoir by the first baffle. The flow separates off the end of each baffle, giving rise to regions of recirculation within which the residence time remains high (**Kris and Ghawi 2007**).

Conclusions

This work deals with the development a specialized strategy for the simulation of the treatment of potable water in different WTPs tanks. The strategy is based on the CFD code Fluent and exploits several specific aspects of the potable water application**.** The case considered above demonstrates not only how CFD can be used as a design tool to improve process tank efficiency, but it also demonstrates how simple baffles can make a difference. With tools such as CFD it is not a question to baffle or not to baffle, but rather how to baffle.

References

Ghawi A. Hadi and Jozef Kris (2007a).” **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

Ghawi A. Hadi, and Jozef Kris (2007b) “**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.

Ghawi A. Hadi and Jozef Kriš (2008a). **“Analysis of the Effect of Baffles Number on Contact Tank Efficiency**”. International Doctoral Seminar 2008, Smolenice Castle, 18 - 20 May 2008 Slovakia. pp. 99-105. ISBN 978-80-8096-058-2.

Ghawi A. Hadi and Jozef Kriš (2008b). **“Numerical Modeling of a Contact Tank at Drinking Water Treatment Plant**”. International Doctoral Seminar 2008, Smolenice Castle, 18 - 20 May 2008 Slovakia. pp. 91-98. ISBN 978-80-8096-058-2.

Ghawi A. Hadi, and Jozef Kris (2008c) **“A Numerical Model of Flow in Sedimentation Tanks in Slovakia”** POLLACK PERIODICA, An International Journal for Engineering and Information Sciences. DOI: 10.1556/Pollack.3., Vol. 3, No. 2, pp. 59–73 (2008). www.akademiai.com.

Kris Jozef and Ghawi A. Hadi. (2007) “**Potable Water Service Reservoirs Design Improvement Using Computational Fluid Dynamics**”. 4th IWA Specialist Conference.on Efficient Use and Management Of Urban Water Supply 21-23, May, JejuIsland, Korea. pp. 1225-1227.

Kris Jozef, and Ghawi Ali Hadi (2007), **Improved, Modeling, Simulation and Operational Parameters of Settling Tank**, 6th International Conference, Hungary 12-18 August

Table 1 Performance data for modelled settling tank

Data	Original	Perf. Baffle (1)	Inboard Launder (2)	Perf. Baf. and Inboard Launder
effluent suspended solids (ESS) mg/l	25	10	6	3

(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.

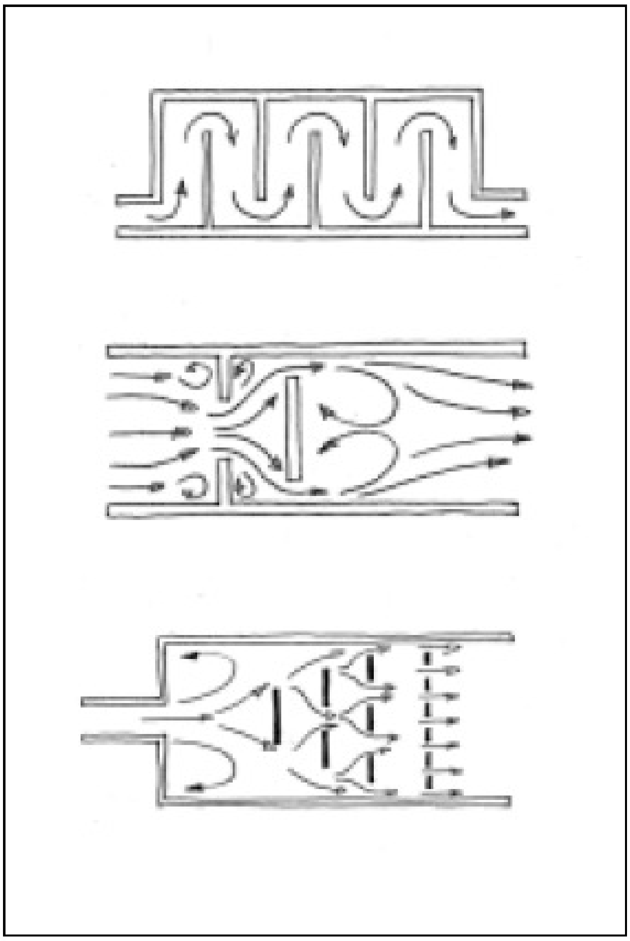

Figure 1a, 1b & 1c (from top to bottom) Different baffle applications.

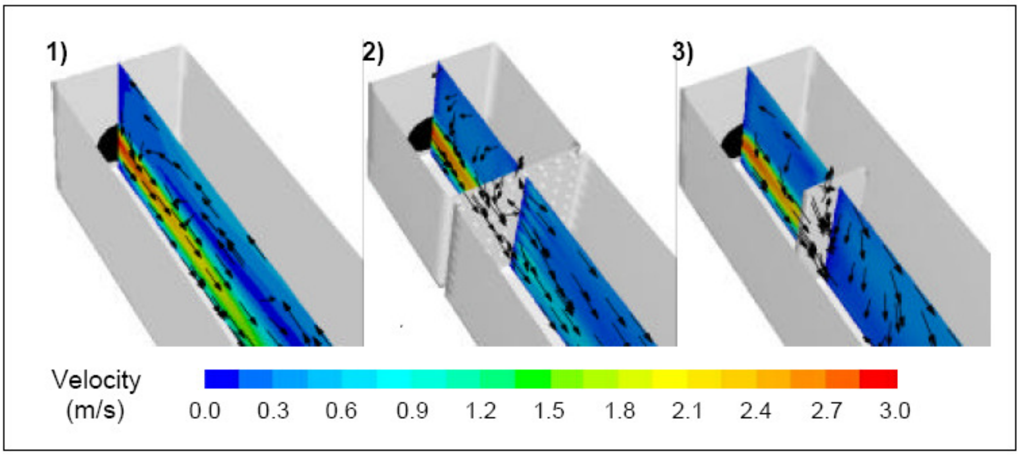

Figure 2 Flow jetting and recirculation, downstream of chlorine contact tank inlet. Mid width velocity profiles for three (3) baffle cases: 1) none (worst); 2) perforated full width; and 3) solid partial width (best). Vectors indicate direction, and contours indicate magnitude (m/s).

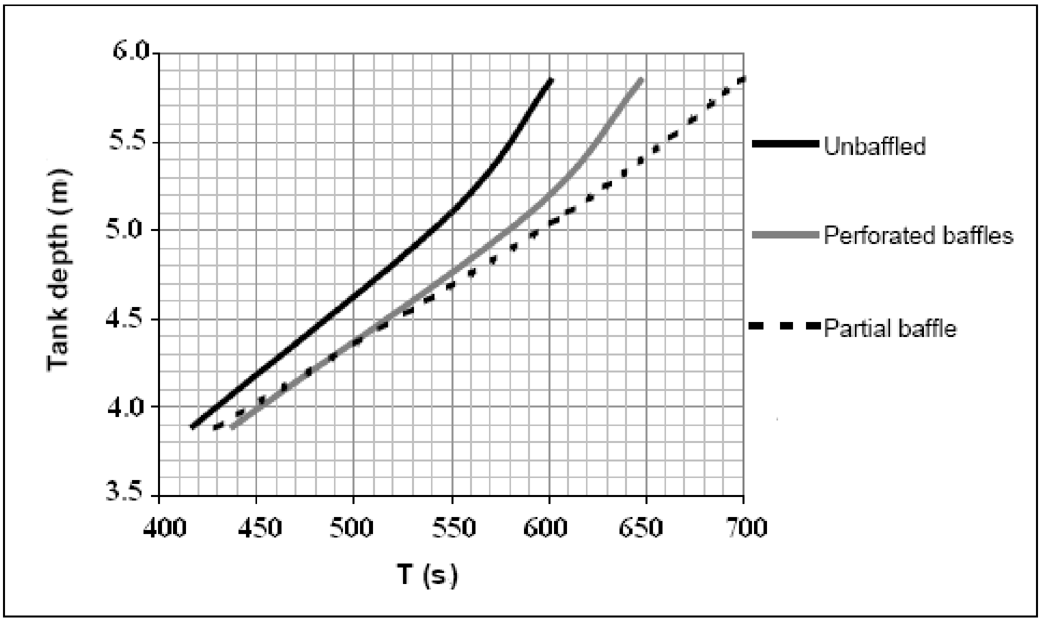

Figure 3 Chlorine contact tank depth required to meet the target t_{10}, for the proposed design, and for tanks with a range of modeled physical interventions.

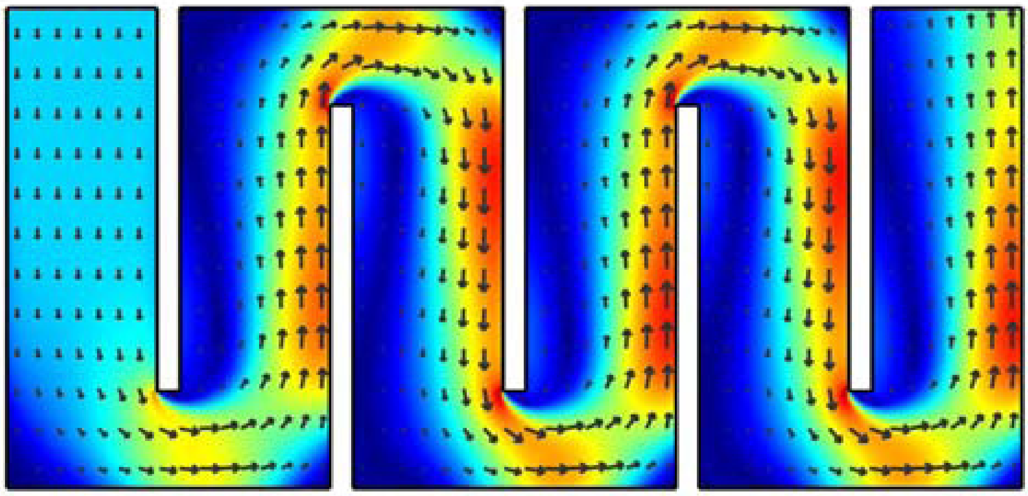

Figure 4 – Highly turbulent zones after baffle tips results in high G-values (red areas). Low G-value areas (blue) are due to straightening of flow after each turn.

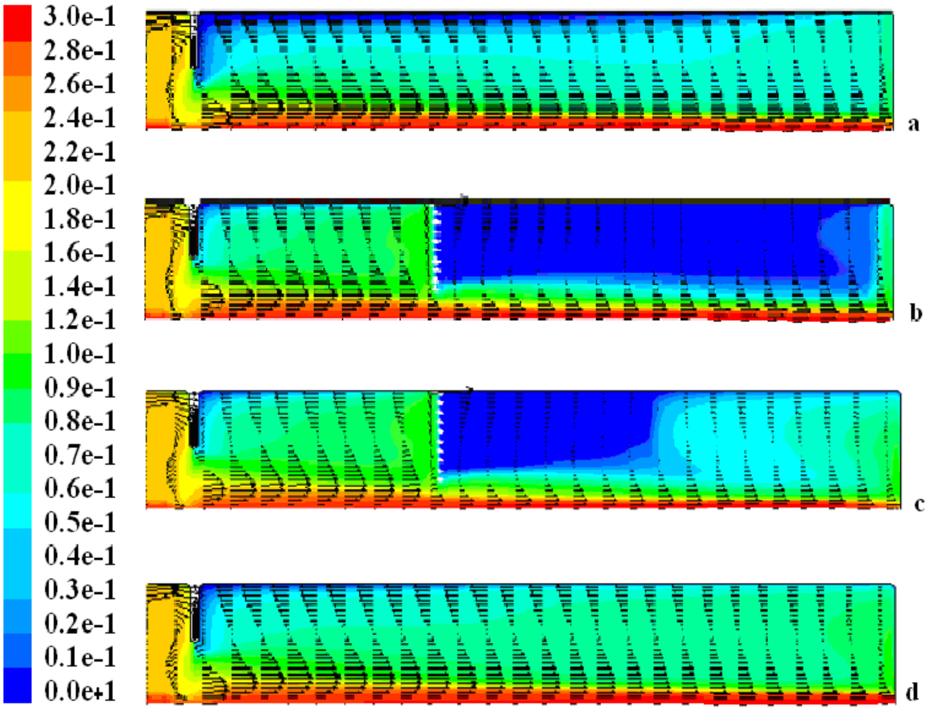

Figure 5 The model simulated velocities and solids concentration (g/l) distributions for the original and modifications tanks (in a vertical section along tank central axis) (a. original tank, b. Perf. Baffle, c. Perf. Baffle and Inboard Launder, and d. Inboard Launder

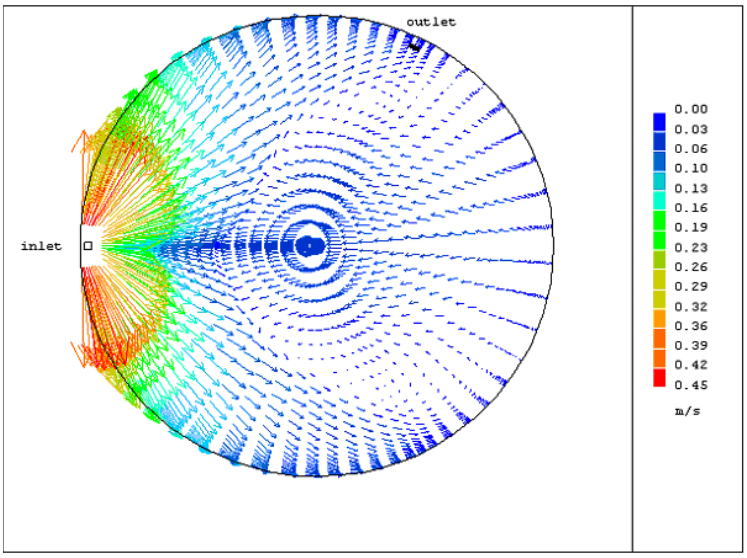

Figure 6 Velocity magnitude vectors at floor (m/s).

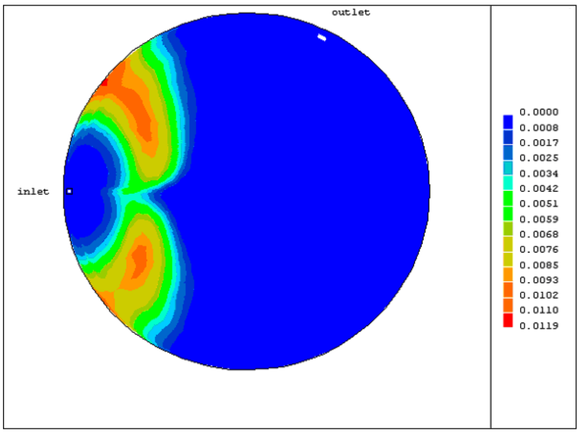

Figure 7 Profiles of tracer contours at floor the tank from the CFD model, 1 minute after tracer injections.

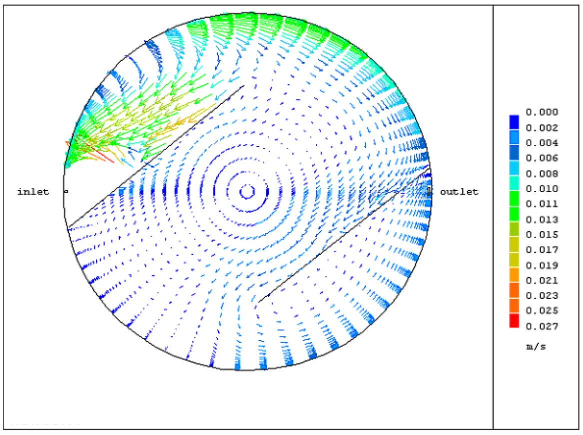

Figure 8 Velocity magnitude vectors at surface (m/s) (with baffles).

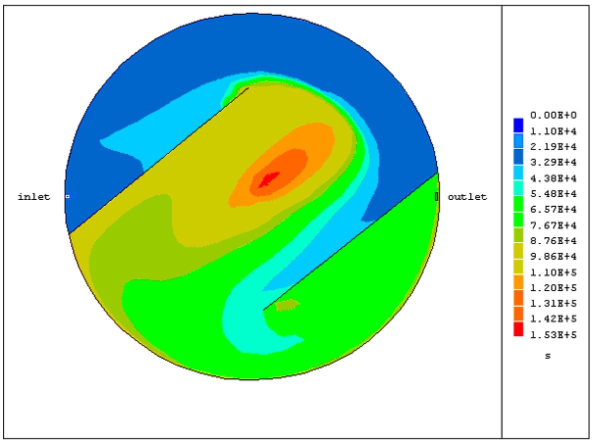

Figure 9 Profiles of tracer contours at surface the tank from the CFD mode (with baffles).