Journal of Applied Fluid Mechanics, Vol. 1, No. 1, pp. 43-54, 2008.
Available online at www.jafmonline.net, ISSN 1735-3645.

Determination of Settling Tanks Performance Using an Eulerian-Lagrangian Method

A.Tamayol[1], B. Firoozabadi[1] and G. Ahmadi[2]

[1]*School of Mechanical Engineering, Sharif University of Technology, Tehran, 11365-8639, Iran*
[2]*Department of Mechanical and aeronautical Engineering, Clarkson University, Potsdam, NY 13699, USA*

Email:firoozabadi@sharif.edu

(Received August 16, 2006; accepted January 21, 2007)

ABSTRACT

Circulation regions always exist in settling tanks. These regions reduce the tank's performance and decrease its effective volume. The recirculation zones would result in short-circuiting and high flow mixing problems. The inlet position would also affect the size and location of the recirculation region. Using a proper baffle configuration could substantially increase the performance of the settling tanks. A common procedure for the comparison of the performances of different tanks has been using the Flow Through Curves (FTC) method. FTC, however, neglects tendencies for particles sedimentation. In this work, a new method for evaluation of the settling tanks performance is presented. The new method which is referred to as the particle Tracking Method (PTM) is based on an Eulerian-Lagrangian approach. In this paper, by using FTC and PTM the effects of the inlet position and the baffle configuration on the hydraulic performance of the primary settling tanks were studied and results were compared. Then, shortcomings of the FTC approach werestated. The optimal positioning of the baffles was also determined though a series of computer simulations.

Keywords: Settling Tanks, Particle Tracking Method (PTM), Flow Through Curve (FTC), Baffle configuration.

NOMENCLUTURE

C	Concentration	t	Non-dimensional time
C_0	Average concentration of the tank	u	Horizontal velocity
$C_{1\varepsilon}, C_{2\varepsilon}$	Constants in turbulence model	v	Vertical velocity
H	Baffle height	Vs	Settling velocity
h	Depth of the tank	ε	Dissipation rate
k	Turbulent kinetic energy per unit mass	ν	kinematic viscosity
Q	Inlet flow rate	ν_t	Turbulent viscosity
T_{Th}	Theoretical detention time	σ_c	Turbulent Schmidt number

1. INTRODUCTION

Sedimentation by gravity is one of the most common approaches for the removal of suspended solid particles from water in refinery plants. In particular, settling tanks are important components of the water treatment plants. The performance of these tanks would directly or indirectly affect the rest of the plant. Therefore, the design of tanks with a high deposition rate and high hydraulic efficiency is critical and has been the subject of many theoretical, experimental and numerical investigations.

Depending on the use of coagulants, the sedimentation basins are divided into two main categories. The primary tanks without using coagulants have relatively low large particles concentration. The flow field in these tanks is not influenced by the particle concentration and the buoyancy effects are negligible. The disinfection tanks also have the same characteristics. By using coagulants in the secondary or final settling tanks, the large particles concentration is generally high and the flow field is affected by the particle concentration.

For a tank with a high performance, it is essential to have a uniform and rather calm flow field. In general, however, the recirculation zones always appear in the settling tanks. The presence of these regions may have different effects. These dead zones would reduce the effective volume of the tanks. The recirculation may cause short circuiting between the inlet and outlet of the tank. In this case, some of the influent would exit the tank without losing their particular content to sedimentation. Furthermore, the recirculation creates regions with high turbulence intensity, which would not only reduce the particle sedimentation, but also may cause the particle resuspension problem.

Current design procedure of the settling tanks is mostly based on the detention time and the assumption of uniform flow and the uniform settling particle velocity. These assumptions that neglect the hydrodynamics of the tank are unrealistic and could result in an improper tank design.

There are some ways to reduce the size of these dead zones which would increase the tank performance. One is to use suitable baffle configurations. It must be emphasized that the use of baffles without sufficient investigation could result in the tanks with worse performance than the ones without baffle. The baffles implementation cost is rather high. Therefore, it is essential to study the optimal position and size of the baffles for the specific settling tanks.

Determination of the flow and concentration fields in the tank can be achieved by using either experimental or theoretical/computational means. Experimental investigation of the flow field is normally time-consuming and expensive. Thus, the simplified theoretical models have often been used. Although the early researchers like Dobbins (1944) and Camp (1946) were aware of the importance of turbulent mixing and recirculation zones, they were not able to provide adequate solutions due to lack of suitable hydrodynamics and turbulence models.

More detailed studies presented by Larsen and Gotthardson (1977) created a new area of investigations in the recent years. Imam *et al.* (1983) solved the flow equations with a constant turbulent eddy diffusivity assumption. Celik *et al.* (1985) and Adams *et al.* (1990) used the $k-\varepsilon$ turbulence model to predict the turbulent flow field in the settling tanks. They used Flow Through Curves (FTCs) for predicting the performance of the settling tanks. Stamou *et al.* (1990) investigated the discretisation accuracy effects on the flow field and FTCs.

Stamou *et al.* (2001) used CFD for increasing the performance of a real settling tank. They used FTC for investigating the performance changes for different baffle configurations. Ashjari and Firoozabadi (2003) used the nonlinear $k-\varepsilon$ and FTC for prediction of flow and tanks performance. Using FTC Tamayol et al. (2005) studied the effects of different inlet positions on the flow field and the efficiency of the settling tanks. Firoozabadi et al. (2005) used FTC for determination of the suitable baffle position in a rectangular settling tank.

FTC neglects the sedimentation effects of particles on the performance of the settling tanks. In this paper, an Eulerian-Lagrangian simulation procedure and a particle tracking method (PTM), is used for prediction of the tanks performance. In this method, the effects of gravitational sedimentation are taken into account. Suitable baffle and inlet positions are determined with the use of the two performance methods and the results are compared. Shortcomings of the FTC method are also discussed. Finally, the suitable baffle configuration for a settling tank configuration is determined.

2. GEOMETRY SPECIFICATION

Two different geometries are used in this work. The effects of different positions of the inlet opening for a tank geometry, which is similar to the settling tank in the city of Sarnia, Ontario, Canada, are studied. The schematic of the tank is shown in Fig. 1. The second geometry is a test model, which is shown in Fig. 2. This tank is located in Karlsruhe, Germany and has been studied by a number of researchers in the past (Stamou et al. (1990)). Studies of the suitable baffle position for this test tank are also performed through a series of computer simulations. Since there was uniformity in width of the tanks, two-dimensional computer simulations for these rectangular tanks are used for simplicity. If the geometry of the tank was so that the uniformity of the flow field in normal direction was not a good approximation, a three dimensional modeling must be used.

3. FTC AND HYDRAULIC EFFICIENCY

Design of the settling tanks is based on the detention time (DT) of particles and the flow in the tank. Theoretical DT of a tank is $T_{TH} = V / Q_D$ in which Q_D is the volume flow rate of the tank and V is the volume of the tank. Hydraulic efficiency is also related to this DT of the tank. If DT is greater than the time needed for particles to settle to the bed of the tank, then particles would settle down in the tank; otherwise, the particles will exit the tank with the effluent. Real DT of a tank (T_{Re}) is lower than T_{TH} one. This is because of the deviation from of the velocity profile form the plug flow condition.

For determination the performance of a tank, two methods are used. One is to determine the concentration field everywhere in the tank, and the other one to use the Flow Through Curves (FTC) method. The FTC method was first used in experimental investigations but as a result of its simplicity in numerical simulations in comparison with solution of complete governing equations considering settling velocity of particles, it became popular. It also provides important information about the mixing condition and the degrees of short-circuiting of the tank. In this approach, dye or a colored fluid that has the same density as water, is injected at the inlet at a time of about 10% of T_{TH} and its concentration is measured at the outlet and is plotted verses time. For better comparison, the concentration and time axes are non-dimensionalized by C_0, which is the mean concentration of the tank, $C_0 = M_{in}/V$, and T_{TH}, respectively. Non-dimensional time is shown with (*t*).

Characteristics and parameter of the FTC, which can be used for determination of different aspects of the hydraulic field and efficiency, are divided into the following categories.

1- Parameters that can be used for investigation of short-circuiting problem, are t_0 and t_{10}. Indices are the percentage of the injected material which passed through the outlet. Higher values of t_0 and t_{10} mean less possibility of short circuiting.

2- Parameters for determining the degree of mixing are measured from the width of the curve. These parameters are $t_{75} - t_{25}$, $t_{90} - t_{10}$ and t_{90} / t_{10}. When these values are higher it means the flow in the tank is highly mixed, which it is not desirable.

3- For prediction of the performance, t_{50} and time are used when the concentration at the outlet is maximum (t_{max}). Higher values of these parameters mean higher performance of the tanks. It must be noted that the best way for prediction of the efficiency is to determine the concentration field.

4. PTM AND SEDIMENTATION PERFORMANCE

As it was stated before, the sedimentation characteristics of the suspended particles are not taken into account when FTC is used. In the new method, particles with realistic densities are injected into the inlet and they are tracked until they are either escape the through the tank outlet or they are trapped at the tank bottom. Efficiency of the tank (η) is related to the capture efficiency (the number of particles trapped at the bottom: N_t).

$$\eta = \frac{N_i - N_o}{N_i} = \frac{N_t}{N_i} \qquad (1)$$

where Ni is the number of injected particles and No is the number of escaped particles. Since this approach considers realistic particles, the simulation results are expected to be more realistic.

It must be noted that this method is suitable for diluted flows and flows with small particles such as the flow in primary settling tanks. For final settling tanks which have large coagulation of particles, the particle interactions and interactions of the fluid and particles must be considered.

5. GOVERNING EQUATIONS

The governing equations are the conservation of mass and momentum. For closing the momentum equations, turbulence models for calculation of Reynolds Stresses are added to these equations.

The mass conservation equation for water (fluid) is:

$$\frac{D\rho}{Dt} + \rho(\nabla . V) = 0 \qquad (2)$$

For a two dimensional incompressible flow, the momentum equations are:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\rho \partial x} + \upsilon\frac{\partial^2 u}{\partial x^2} + \upsilon\frac{\partial^2 u}{\partial y^2} + \frac{\partial}{\partial x}\left(\upsilon_t \frac{\partial u}{\partial x}\right) + \frac{\partial}{\partial y}\left(\upsilon_t \frac{\partial v}{\partial x}\right) \qquad (3)$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\rho \partial y} + \upsilon\frac{\partial^2 v}{\partial x^2} + \upsilon\frac{\partial^2 v}{\partial x^2} + \frac{\partial}{\partial x}\left(\upsilon_t \frac{\partial u}{\partial y}\right) + \frac{\partial}{\partial y}\left(\upsilon_t \frac{\partial v}{\partial y}\right) \qquad (4)$$

These equations are written with the Bousinesq approximation. For modeling of turbulence, the RNG $k-\varepsilon$ model is used. Tamayol and Firoozabadi (2006) have shown that the RNG $k-\varepsilon$ model can capture the curvature streamlines better than the standard $k-\varepsilon$ model. The corresponding transport equations for turbulent kinetic energy (k) and its dissipation rate (ε) are:

$$\rho\frac{Dk}{Dt} = \frac{\partial}{\partial x_i}\left[\alpha_k \mu_{eff} \frac{\partial k}{\partial x_i}\right] + G_k + G_b - \rho\varepsilon - Y_M \qquad (5)$$

$$\rho\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i}\left[\alpha_k \mu_{eff} \frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b) - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} - R \qquad (6)$$

In these equations, G_b, Y_M and G_k are production of turbulence kinetic energy due to buoyancy, compressibility and mean velocity gradients, respectively. The first two, here are zero but the third is

$$G_k = -\rho\overline{u_i' u_j'}\frac{\partial u_j}{\partial x_i} \qquad (7)$$

Model constants are: $C_{1\varepsilon}=1.42, C_{2\varepsilon}=1.68$. Although the turbulent viscosity can be computed from a differential equation, an algebraic relation is used. i,e.,

$$\mu_t = \rho C\mu \frac{k^2}{\varepsilon} \text{ , } C\mu = 0.0845 \qquad (8)$$

In Eq. 6, R is defined as

$$R = \frac{C_\mu \rho \eta^3 (1-\eta/\eta_0)}{1+\beta\eta^3}\frac{\varepsilon^2}{k} \qquad (9)$$

$$\eta = \frac{Sk}{\varepsilon} \qquad \eta_0 = 4.38 \qquad \beta = 0.012$$

$$S = \left[\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\right]^{\frac{1}{2}} \qquad (10)$$

The particle concentration equation is:

$$\frac{\partial C_i}{\partial t}+\frac{\partial uC_i}{\partial x}+\frac{\partial vC_i}{\partial y}=\frac{\partial}{\partial x}\left(\frac{\upsilon_t}{\sigma_c}\frac{\partial C_i}{\partial x}\right)+ \frac{\partial}{\partial y}\left(\frac{\upsilon_t}{\sigma_c}\frac{\partial C_i}{\partial y}+V_{Si}C_i\right) \qquad (11)$$

Here C is the concentration and σ_c is turbulent Schmidt number which is taken equal to 1 (Stamou et al. (1990)).

In Eq. (11), V_s stands for the terminal settling velocity of particles that needs to be evaluated. In the primary settling tanks a discrete model is used for V_s. For calculation of FTC, V_s is set equal to zero.

In PTM method, the flow field is obtained from the solution of Eqs. (1)-(10), but Eq. (11) is not used. Instead, equations of motion for particles are solved. That is,

$$\frac{du_p}{dt} = F_D(u-u_p)+\frac{g(\rho_p-\rho)}{\rho_p}+F_x \qquad (12)$$

In which Re is the particle's Rynolds number.

$$F_D = \frac{18\mu}{\rho_p d_p^{\ 2}}\frac{C_D\,\boldsymbol{Re}}{24} \qquad (13)$$

$$\boldsymbol{Re} = \frac{\rho d_p |u_p-u|}{\mu} \qquad (14)$$

$$C_D = \frac{24}{\text{Re}}\left(1+0.15\,\text{Re}^{0.687}\right) \qquad (15)$$

In Eq. (12) drag and buoyancy forces are present. Since the size of particles in settling tanks is small, lift and Brownian forces are not considered so the last term is equal to zero. u_p is the velocity of particles.

u in Eq. (12) is the instantaneous velocity of fluid which is the sum of the mean and fluctuation velocities. Mean velocity is known from the solution of the flow field (Eqs. 1-10). For modeling of fluctuations an eddy life time model is used. Accordingly,

$$u' = \zeta\sqrt{\overline{u'^2}} = \sqrt{2k/3} \qquad (16)$$

where ζ is selected from a population with a Gaussian distribution with zero mean and unit variance.

It must be noted that since the instantaneous velocities are random in each calculation results are different from previous computations and a stochastic procedure is used and each case are calculated 30 times and mean values are chosen for PTM results.

6. BOUNDARY CONDITIONS AND SOLVERS

Symmetry condition is used for the free surface boundary condition. In the present work, the effects of the wind and small ripples on the flow-field are neglected. Values of all properties are known at the inlet, at the outlet values are interpolated from the computational domain.

Walls are treated as non-penetrative boundaries. No-slip condition for velocity and zero concentration gradient in concentration equation are used. Standard logarithmic wall functions are used for turbulence modeling.

In PTM when a particle collides with the bottom surface, it is assumed to be trapped, but for other walls and the free surface it is assumed to be reflected.

A finite volume approach is used for the solution of the governing equations. Second order upwind scheme is selected to discretize the governing equations. SIMPLEC algorithm is used for pressure-velocity coupling. Finally, a CFD code was developed to solve the governing equations with the specified boundary conditions.

7. RESULTS AND DISCUSSION

A settling tank with the simplified geometry of the tank which exists in Sarnia, Ontario is picked for studying the effects of different inlet positions. The inlet of the Sarnia tank is at the surface. Three different positions for the inlet opening are assumed in the present computations. The flow Reynolds number, based on the depth of the tank is 62,000. Different computational grids are used for calculations, and finally a grid with the size of 600×160 is found whose results have acceptable independency from the grid.

In Fig. 3 the predicted velocity are compared to experimental and numerical ones available in (Stamou et al. (1990)). The reasons of the difference between experimental and numerical results are described in (Stamou et al. (1990)). The most important reasons are simplifications which are used for the geometry modeling and lack of accuracy in instruments used for measuring the velocity in the real tank

For clarity, in Figs. 4 - 6 only circulation zones for different values of S_i / h are shown instead of the entire flow-field. When the inlet is at the surface of the tank (Fig. 4, $S_i / h = 0.75$), one recirculation region with a length of about 9.3 m exists which occupies 10% of total volume of the tank. In Fig. 5, two vortices are seen with maximum length of 4.2 m for $S_i / h = 0.5$, which spoils only 5% of total volume of the tank. For $S_i / h = 0.25$, which is shown in Fig. 6, one circulation zone with a length of 8.1 that occupies 9% of the total volume appears. From these observation, it is conjectured that the case with $S_i / h = 0.5$ would have a better performance

In Fig. 7, the FTC diagrams for different values of S_i / h are shown. T_{TH} of this tank is 1782.6 seconds. Different important values of FTCs are listed in Table 1. It can be seen that $S_i / h = 0.5$ has higher values of t_0 and t_{10} which means that short-circuiting is lower in this case. Values of $t_{75} - t_{25}$, $t_{90} - t_{10}$ and t_{90} / t_{10} which show the degree of flow mixing are lower than in other cases.

Finally higher values of t_{50} and t_{max} show a higher hydraulic efficiency. This result was also predictable from the flow-field and streamlines. The inlet position with $S_i / h = 0.5$ appears to be the optimal case. The case with $S_i / h = 0.25$, which is the inlet from the bottom of the tank, has a better performance than those the case with the top inlet

These cases are also studied with PTM. Three different sizes for particles are used and the efficiency of the tank is determined. Table 2 shows when the inlet is located near the bottom, a higher efficiency is obtained. The reason is that when the inlet is near the bottom, particles have a shorter distance to reach the bottom.

Another way for increasing the performance of a settling tank is to use a baffle in the tank. In this part, the second geometry shown in Fig. 2 is used and S_i / h is set equal to 0.91 which means that the inlet opening is placed near the surface. Reynolds number is equal to 10000 based on the inlet opening height. Different grids were used for calculations and finally a 75×35 grid was selected for computation of results.

Six different positions and two different sizes for baffles are used for this part. The positions of the baffles are determined with respect to the streamlines of the same tank without any baffle (WB). Streamlines of the tank without baffles are shown in Fig. 8. This figure shows that baffles are placed in three distances from the inlet. First, near the inlet and near the circulation zone at x = 0.35 m. Second, near the outlet at x = 1.8 m and third, at x = 0.9 m which is near the middle of the tank. Baffles are placed on the bottom or attached to the surface of the tank. These geometries are shown in Fig 9. As it is shown in Fig. 8, the reattachment point is about 0.66m, which is close to the experimental value of 0.7m (Stamou et al. (1990)). When we use a baffle in the tank two circulation regions would appear; one under the inlet of the tank, the same as the case WB and the other behind the baffle. The size of both circulation zones is related to the position and height of the baffle. In this investigation baffle heights are taken 1.5 and 2.5 times of the inlet opening.

For increasing the performance of settling tanks another method has been suggested, which is to use the grid baffle near the inlet. This kind of baffle may result in a more uniform distribution of velocity in the tank. Two positions for grid baffle are shown in Fig. 10.

Total volumes of the two dead positions mentioned before are presented in Table 3. These values must be compared with the volume of 567 cm^2 occupied by the vortex exists in the tank without baffle

It is seen that when the baffle is placed near the circulation zone, the total size of the circulation region is reduced. The best position appears to be in the middle of the circulation region, because at this position, the baffle cuts the circulating streamlines. For better judgment the streamlines for baffle with position 3 and height 3cm are shown in Fig. 11. Using baffles without good experience and knowledge would increase the size of circulation zones. Because of this problem in primary settling tanks, baffles are not used.

FTC of cases having baffles with the height of 3 cm is shown in Fig. 12. From Table 3 it is predictable that some cases must have poor performance. This is related to the size of the dead zone. As FTC diagrams are not clear enough, Table 4 which contains important data from FTC is presented.

t_0 and t_{10} are important to understand the degree of short-circuiting. Higher values mean a lower degree of short-circuiting. Between these cases, case 1 is better than others from this point of view but case of without baffles (WB) and case three are also acceptable. Case 5 is acceptable for this part. t_{max} and t_{50} are indicators of efficiency, so if they are high, it means hydraulic efficiency is high also. Case 1 and 3 are much better than others in comparison with the case WB. When two different values which appeared in the last columns are lower, it means that the flow is more uniform. From this point of view, case 3 is also better than the others. According to Table 4, grid baffles have better conditions and they will improve the flow field and an increase in performance is predicted regarding to FTC predictions.

PTM is used for determination of suitable baffle shape and position. Results are presented in Table 5.

It is obvious that the baffle position 1 is the best position and the basin with baffle position 1 has even a higher efficiency than the grid baffle. Figure 13 shows the streamlines of the tank with a baffle in position 1. If Fig. 13 is considered it is seen that the particles are forced to move toward the bottom so that particles have less distance from the bottom after the baffle.

Results obtained from the analyses of suitable inlet position and suitable baffle position showed that FTC and PTM are not in complete agreement. It is due to having neglected the sedimentation of particles which are moving toward the bottom as a result of their weights. FTC is suitable for analysis of disinfection tanks in which sedimentation has not occurred. PTM is very realistic and can be used for modeling of settling tanks. It must be noted PTM is also acceptable only for tanks with low concentration.

Finally, the use of a reflection entrance baffle in such a way that make the flow change its direction towards the bottom of the tank is advised for primary settling tanks.

8. CONCLUSIONS

The effects of different baffle positions and different baffle configurations on the performance of settling tanks were studied. A new method, PTM was proposed for determination of performance of primary settling tanks. In the present work results are obtained using two different methods. FTC method which is common for the analysis of the primary settling tanks and disinfection basins was used and the results were compared with those obtained from PTM. FTC neglects the sedimentation of the particles but PTM considers sedimentation in its modeling. Results were not in agreement and it showed that FTC is not suitable for predicting the performance of primary settling tanks and it is acceptable for the analysis of disinfection tanks. The best position for the inlet is near the bottom and existence of a reflection entrance baffle near the free surface of settling tanks can increase the performance of primary settling tanks.

REFERENCES

Adams, E.W. , Rodi, W. (1990),` Modeling flow and mixing in sedimentation tanks`, J. Hydraulic Engineering, Vol. 116, No. 7, pp. 895-913.

Ashjari, M.A. and Firoozabadi, B. (2003), `Use of nonlinear $k-\varepsilon$ in calculation of performance of settling tanks.`, Int. Conf. Mech. Engineering, Mashhad, Iran.

Camp, T.R., (1946),` Sedimentation and the design of settling tanks`, Transactions, ASCE, Vol.3 PP. 895-936.

Celik, I., Rodi, W. and Stamou, A.I. (1985),` Prediction of hydrodynamic characteristics of rectangular settling tanks`, Int. Symposium of Refined Flow Modeling and Turbulence Measurements, Iowa USA.

Dobbins, W.E. (1944), `Effects of turbulence on sedimentation `, Transactions, ASCE, Vol. 109, no. 2218, pp. 629-656.

Firoozabadi B., Tamayol A. and Ahmadi G. (2005), `Effects of inlet position and baffle configuration on the hydraulic performance of primary settling tanks` ASME IMECE, Florida, USA.

Imam, E. and Mc Corquodale, J.A. (1983),` Numerical modeling of sedimentation tanks `,Proc. ASCE Vol. 109,Vol. 109,No.HY 12, pp.1740 -1754

Larsen P., (1977), “On the Hydraulics of Rectangular Settling Basins-Experimental and Theoretical Studies”, Report No. 1001, Department of water Resources Engineering, Lund, Sweden.

Stamou, A.L., Adams, E.W. and, Rodi, W. (1990),` Numerical modeling of flow and settling in primary rectangular Clarifiers`, J. Hydraulic Research, vol.27, pp. 665-682.

Stamou, A., Katsiris, I., Georgiadis, S., Kanellopoulou, S., Tzoumerkas, F. and Xenos, D. (2001), `Improving the efficiency of existing water process tanks using water process tanks using flow through curves (FTCs) and mathematical models`, Int. Conf. on Environmental Science and Technology, Greece.

Tamayol, A., and Firoozabadi, B.,. (2006), ` Effects of turbulent models and baffle position on hydrodynamics of settling tanks`, Scientia Iranica Vol. 13, No. 3, PP 255-260.

Tamayol, A., Nazari, M., Firoozabadi, B. and Nabovati, A. (2005), `Numerical modeling and study of effects of inlet position and height of inlet baffle on the performance of settling tanks. `, Fluid Dynamics Conf., Iran (In Farsi).

Table 1- Important values from FTC for first geometry.

	t_0	t_{10}	$t_{75}-t_{25}$	$t_{90}-t_{10}$
$S_i/h=0.75$	0.56	0.70	0.32	0.72
$S_i/h=0.5$	0.68	0.84	0.17	0.43
$S_i/h=0.25$	0.61	0.78	0.20	0.51
	t_{90}/t_{10}	t_{max}	t_{50}	
$S_i/h=0.75$	2.03	0.75	0.88	
$S_i/h=0.5$	1.51	0.91	0.95	
$S_i/h=0.25$	1.65	0.85	0.89	

Table 2- Results obtained from PTM for different inlet positions (mean values)

Particles Diameter	30 μm		
Status	N_{ti}	N_i	η
$S_i/h=0.75$	997	1003	0.5
$S_i/h=0.5$	1055	1045	0.51
$S_i/h=0.25$	1472	1015	0.59
Particles Diameter	50 μm		
Status	N_{ti}	N_i	η
$S_i/h=0.75$	1814	186	0.91
$S_i/h=0.5$	1985	115	0.94
$S_i/h=0.25$	2411	89	0.96

Table 3- Total dead volume for various positions and heights.

position – Baffle height	1-3	1-5	2-3
Total circulation volume	555	638	648
position – Baffle height	2-5	3-3	3-5
Total circulation volume	808	476	483
position – Baffle height	4-3	4-5	5-3
Total circulation volume	716	882	622
position – Baffle height	5-5	6-3	6-5
Total circulation volume	710	590	655

Table 4- Useful data from FTC Diagrams

Case	t_0	t_{max}	t_{50}	$t_{75}-t_{25}$	$t_{90}-t_{10}$
1-3	0.565	0.867	0.97	0.31	0.64
1-5	0.551	0.844	0.96	0.34	0.691
2-3	0.495	0.851	0.95	0.33	0.704
2-5	0.46	0.793	0.92	0.4	0.858
3-3	0.533	0.88	0.96	0.28	0.662
3-5	0.538	0.876	0.96	0.29	0.673
4-3	0.476	0.805	0.93	0.38	0.8
4-5	0.425	0.767	0.92	0.43	0.9
5-3	0.54	0.867	0.95	0.3	0.667
5-5	0.505	0.842	0.95	0.34	0.732
6-3	0.509	0.873	0.96	0.3	0.68
6-5	0.484	0.86	0.95	0.32	0.707
Without baffle	0.55	0.882	0.96	0.28	0.651
Grid baffle d=20	0.64	0.92	0.99	0.22	0.48
Gird baffle d=40	0.63	0.88	0.96	0.26	0.57

Table 5- Results obtained from PTM for different baffle configurations (mean values).

Particles Diameter	56 μm		
Status	N_{ti}	N_i	η
WB	517	283	0.65
Baffle 1	568	232	0.71
Baffle 2	551	249	0.69
Baffle 3	549	251	0.69
Baffle 4	526	274	0.66
Grid baffle x=20	509	291	0.64
Grid baffle x=40	489	311	0.61
Particles Diameter	66 μm		
Status	N_{ti}	N_i	η
WB	607	193	0.76
Baffle 1	638	162	0.8
Baffle 2	626	174	0.77
Baffle 3	607	193	0.76
Baffle 4	609	191	0.76
Grid baffle x=20	589	211	0.74
Grid baffle x=40	565	235	0.71
Particles Diameter	76 μm		
Status	N_{ti}	N_i	η
WB	669	131	0.84
Baffle 1	681	119	0.85
Baffle 2	659	141	0.82
Baffle 3	651	149	0.81
Baffle 4	651	149	0.81
Grid baffle x=20	674	126	0.84
Grid baffle x=40	642	158	0.8

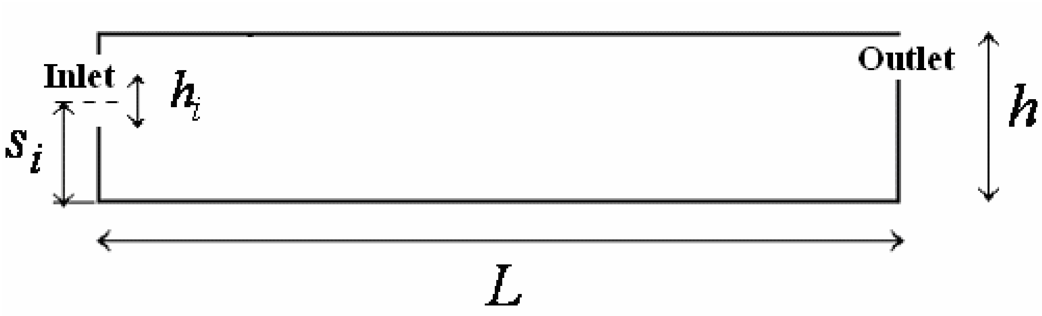

Fig. 1- Simplified geometry of the tank in Sarnia with $h_i = 1.35$ *m* and *h*=2.7*m* and *L*=41 *m*

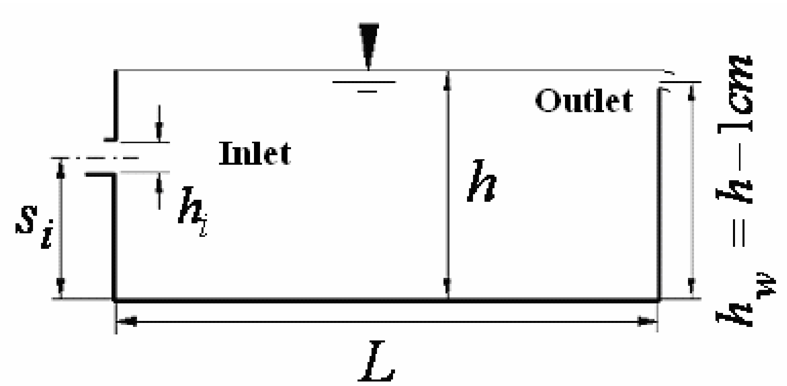

Fig. 2- Geometry of the tank in Karlsruhe with $h_i = 2$ *cm* and *h*=11 *cm* and *L*=250 *cm*.

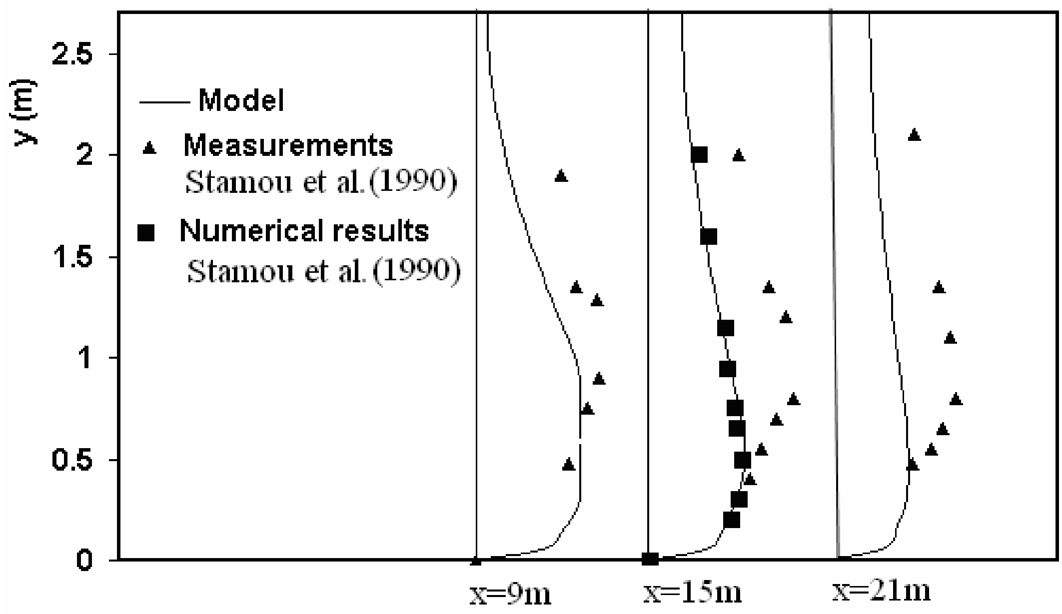

Fig. 3- Comparison of velocity distribution from the present work with former experimental and numerical simulations for simple case for $S_i / h = 0.25$

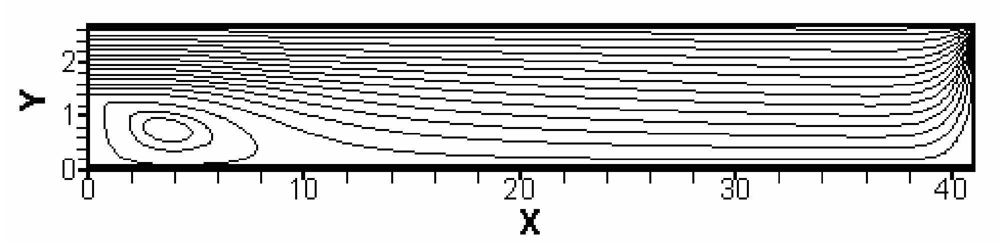

Fig. 4- Streamlines for $S_i / h = 0.75$.

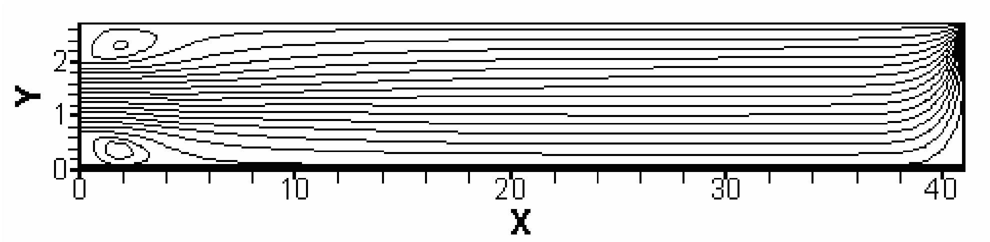

Fig. 5- Streamlines for $S_i / h = 0.5$.

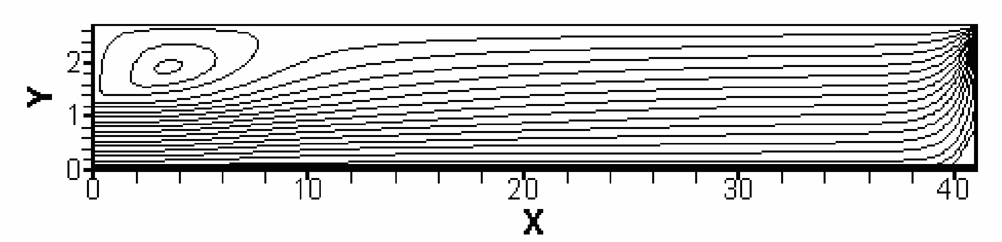

Fig. 6- Streamline for $S_i / h = 0.25$.

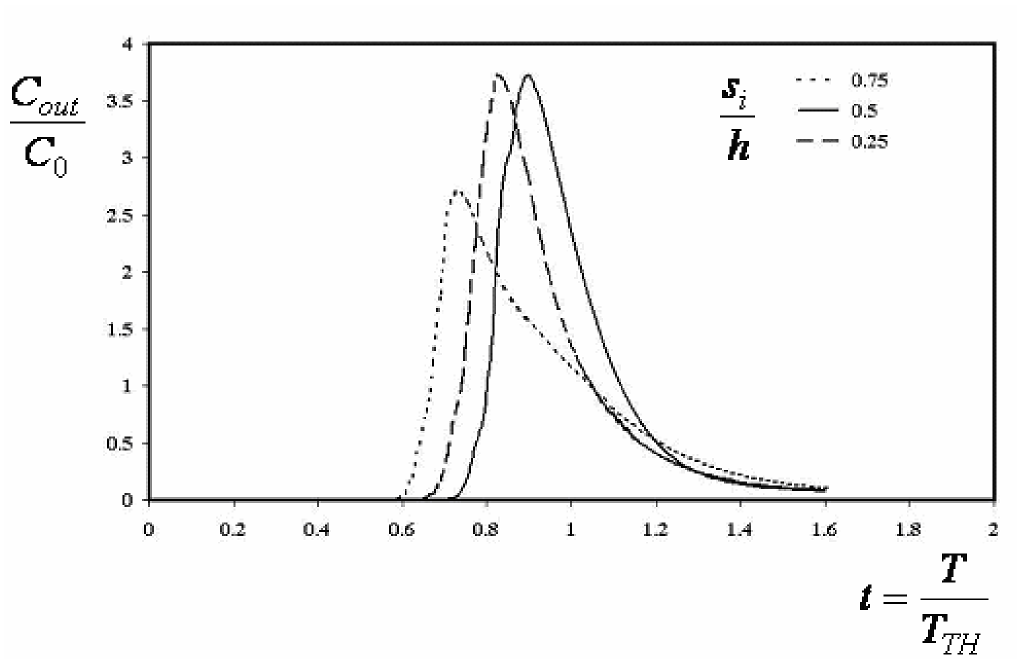

Fig. 7- FTC for different values of S_i / h .

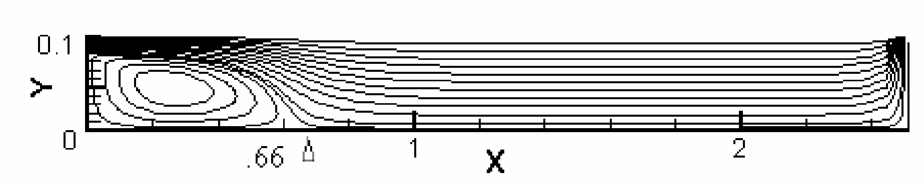

Fig. 8- Streamlines of case without baffle.

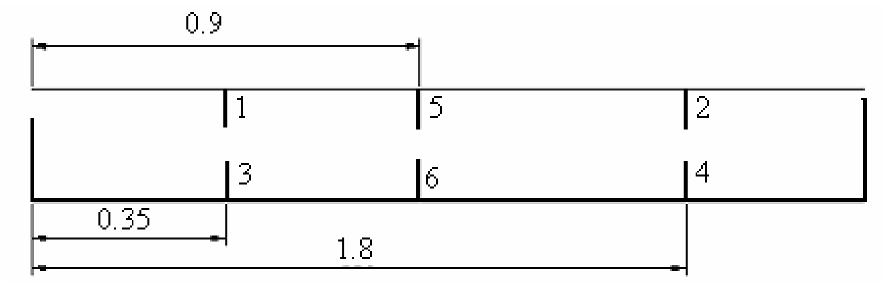

Fig. 9- Different positions of baffles in the tank.

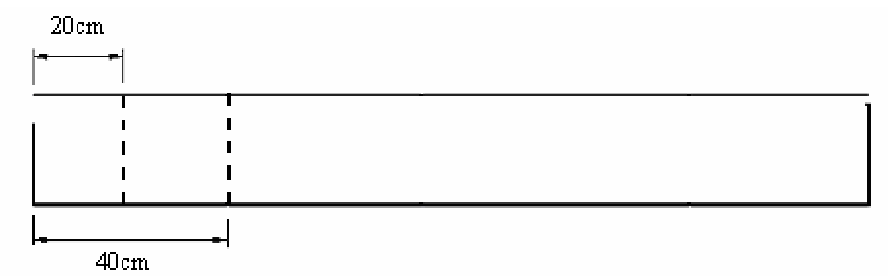

Fig. 10- Different positions of grid baffls in the tank.

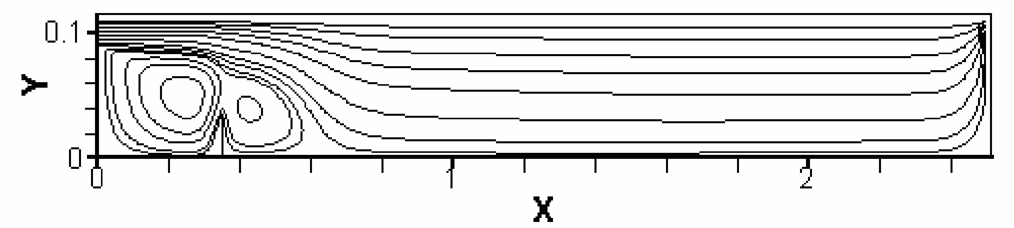

Fig. 11- Streamlines of baffle in position 3 and size 3 cm.

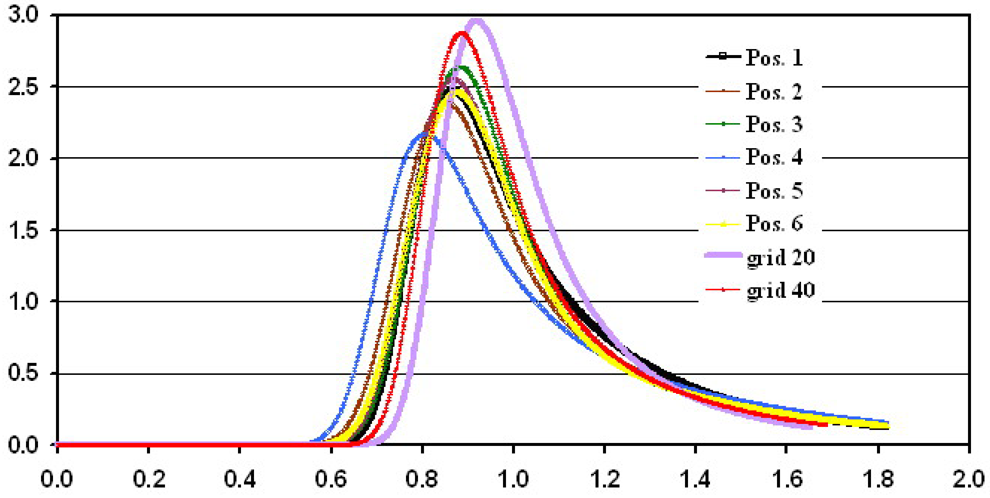

Fig. 12- FTC for different Positions of baffles with height equal to 3.

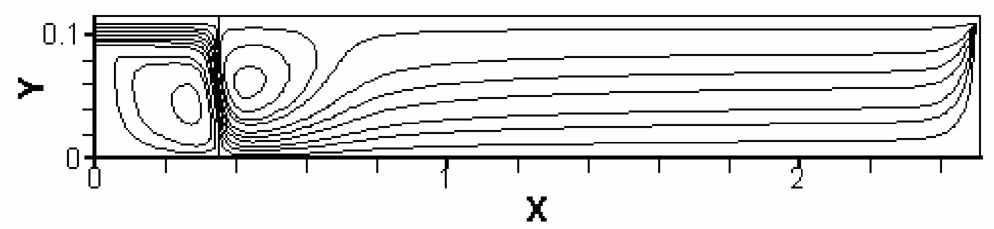

Fig. 13- Stream line for a tank with baffle in position 1.