Scientia Iranica, Vol. 13, No. 3, pp 255–260

SCIENTIA
IRANICA

Effects of Turbulent Models and Baffle Position on the Hydrodynamics of Settling Tanks

A. Tamayol[1] and B. Firoozabadi*

In this paper, the numerical results of hydrodynamic modeling of primary settling tanks are presented. The flow field is assumed to be incompressible and non-buoyant. The effects of two different types of turbulence model, standard $k - \varepsilon$ and RNG, are compared with each other. The effects of an inlet baffle on the hydrodynamics of settling tanks are also studied. Results are obtained for the primary settling tank of the city of Sarnia, Ontario, Canada. The effects of the existence and position of another interior baffle in the settling tanks are also studied. Results in the different parts are compared with experimental and numerical data and showed good agreement. Comparison between two models of turbulence shows that the numerical results of the flow field, especially the streamline curvature, are not the same, in spite of having nearly equal results in the streamwise velocity component.

INTRODUCTION

Sedimentation by gravity is one of the most common and extensively applied treatments for the removal of suspended solids from water and waste water. Investment for settling tanks is about 30% of the total investment for a treatment plant. So, determination of the sedimentation efficiency has been the subject of numerous theoretical and experimental studies.

The sedimentation efficiency depends on the characteristics of suspended solids and flow-field in the tank. Since primary settling tanks have low concentration, flow-field is not much influenced by particles distribution. The flow pattern and the path taken by suspended solids through the tank are closely linked to each other and to the settling tank performance. For example, tanks having short circuiting will display a poorer performance than those with a uniform velocity field without short circuiting. Circulation zones are named as dead zones in tanks because, in these regions, water is trapped and particulate fluid will have less volume for flow and sedimentation. According to this, the existence of large circulation regions will lower tank performance.

Another important parameter is turbulence. The flow in settling tanks is turbulent; therefore, it is of great importance to the concentration distribution prediction and sedimentation of particles. If the turbulence is not predicted correctly, it may cause resuspension of particles which have already settled. Many researchers have worked on the effects of different types of turbulence models on the flow-field and concentration-field. Although the Reynolds number of flow in settling tanks is about 20000-80000, the flow is turbulent. So, turbulence modeling is especially very important in primary tanks. Thus, the assumption of isotropy of turbulence is not a good assumption. In addition, the existence of recirculation zones and the importance of prediction of the size of these regions has made it necessary to use turbulence models carefully. Determination of the flow field can be achieved either experimentally or theoretically by means of mathematical methods. Experimental determination of the flow field is difficult and mostly expensive. For this reason, simplified theoretical models have been used. Although early researchers like Dobbins [1] and Camp [2] were aware of the importance of turbulent mixing and recirculation zones, they were not able to provide adequate solutions, due to the lack of suitable hydrodynamics and turbulence models. More advanced numerical models have been proposed recently by Larsen and Gotthardson [3], Schamber and Larock [4], Imam et al. [5], Abdel-Gawad and McCorquodale [6], Celik et al. [7], Adams et al. [8] and Rodi [9], which have been applied with at least partial success in prediction of the flow field in settling tanks.

1. *Department of Mechanical Engineering, Sharif University of Technology, Tehran, I.R. Iran.*
*. *Corresponding Author, Department of Mechanical Engineering, Sharif University of Technology, Tehran, I.R. Iran.*

Among the above investigators, Imam [5] and Abdel-Gawad et al. [6] have used a constant eddy diffusivity assumption, while Schamber and Larock [4], Celik [7] and Rodi [9] used a $k \quad \varepsilon$ turbulence model. Their results show that concentration prediction depends on the hydrodynamics of the flow field. Most of these researchers concluded that the standard $k \quad \varepsilon$ model is not capable of precisely predicting the size of the recirculation zone, which strongly affects tank efficiency. To overcome the above weakness of the standard model, Adams et al. [8] carried out their calculations with the curvature modification. This model had shown good results in calculating the separation point length of the backward-facing step flow. Nevertheless, the result of overestimating the length of the recirculation zone in the tank was slightly surprising. It seems that in this flow, the formation of streamlines with strong curvature caused a lack in the modified model.

In the present work, two different types of $k \quad \varepsilon$ models have been used for the flow field prediction. It is shown that although the velocity profiles obtained from these models are the same, other parameters, such as eddy viscosity, are different and can cause different streamline patterns.

MATHEMATICAL MODELING

Geometry Specification

As shown in Figure 1, the modeled geometry is a rectangular tank in the city of Sarnia, Ontario, Canada [10]. It is really three dimensional, but, for simplicity, a 2-D model was used, first, by neglecting the bottom slope and the inlet region. The zone between the flow inlet and baffle is called the inlet region. In the second part, the geometry is modeled completely with the inlet region. In the last part, the effects of the interior baffle position are studied.

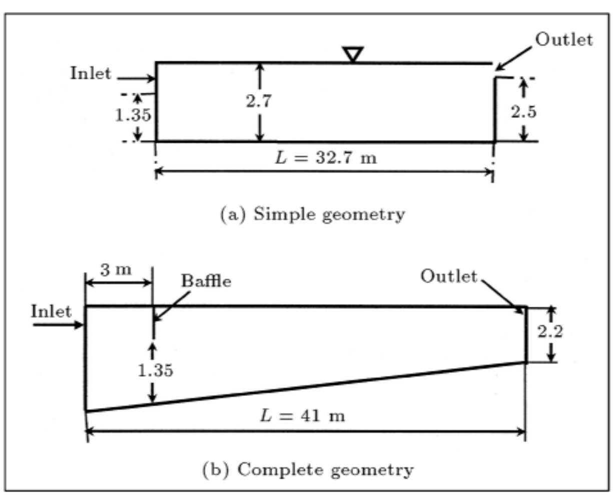

Figure 1. Geometry of problem.

Governing Equations

The flow field is steady, two dimensional, non buoyant and incompressible. The continuity equation in the Cartesian coordinate is written as;

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0. \tag{1}$$

Also, the momentum equations are;

$$u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}= \quad \frac{\partial p}{\rho\partial x}+\frac{\partial}{\partial x}\left(\nu\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\nu\frac{\partial u}{\partial y}\right)+\frac{\partial}{\partial x}\left(\nu_t\frac{\partial u}{\partial x}\right)+\frac{\partial}{\partial y}\left(\nu_t\frac{\partial u}{\partial y}\right), \tag{2}$$

$$u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}= \quad \frac{\partial p}{\rho\partial y}+\frac{\partial}{\partial x}\left(\nu\frac{\partial \nu}{\partial x}\right)+\frac{\partial}{\partial y}\left(\nu\frac{\partial \nu}{\partial y}\right)+\frac{\partial}{\partial x}\left(\nu_t\frac{\partial \nu}{\partial x}\right)+\frac{\partial}{\partial y}\left(\nu_t\frac{\partial \nu}{\partial y}\right). \tag{3}$$

In the present model, the pressure distribution is not assumed to be hydrostatic. The standard $k \quad \varepsilon$ model is a semi-empirical model based on the transport equations for the turbulent kinetic energy (k) and its dissipation rate (ε). The turbulent kinetic energy, k, and its rate of dissipation, ε, are obtained from the following transport equations:

$$\rho\frac{Dk}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_i}\right]+G_k+G_b \quad \rho\varepsilon \quad Y_M, \tag{4}$$

and:

$$\rho\frac{D\varepsilon}{Dt}=\frac{\partial}{\partial x_i}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_i}\right]+C_{1\varepsilon}\frac{\varepsilon}{k}(G_k+C_{3\varepsilon}G_b) \quad C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}. \tag{5}$$

The term G_k, representing the production of turbulent kinetic energy, is modeled identically to the standard $k \quad \varepsilon$ model. From the exact equation for the transport of k, this term may be defined as:

$$G_k= \quad \rho\overline{u_i'u_j'}\frac{\partial u_j}{\partial x_i}. \tag{6}$$

The "eddy" or turbulent viscosity, μ_t, is computed by combining k and ε as follows:

$$\mu_t=\rho C_\mu\frac{k^2}{\varepsilon}, \tag{7}$$

where C_μ is a constant. The model constants have the following default values:

$$C_{1\varepsilon}=1.44, \quad C_{2\varepsilon}=1.92, \quad C_\mu=0.09,$$

$$\sigma_k=1.0, \qquad \sigma_\varepsilon=1.3.$$

In the derivation of the standard $k \quad \varepsilon$ model, it is assumed that the flow is fully turbulent and the effects of molecular viscosity are negligible. The $k \quad \varepsilon$ model is, therefore, valid only for far away from the wall. On the other hand, the turbulence is assumed to be isotropic in the standard $k \quad \varepsilon$ model. Then, it seems that in this kind of flow, at which the Reynolds number is not high enough, the $k \quad \varepsilon$ model may not predict the flow field properly.

The RNG-based $k \quad \varepsilon$ turbulence model is derived from the instantaneous Navier-Stokes equations using a mathematical technique called the "Renormalization Group" (RNG) methods. The analytical derivation results in a model with constants different from those in the standard $k \quad e$ model and additional terms and functions in the transport equations for k and ε. The transport equation for k can be written as:

$$\rho\frac{Dk}{Dt} = \frac{\partial}{\partial x_i}\left[\alpha_k \mu_{\text{eff}} \frac{\partial k}{\partial x_i}\right] + G_k + G_b \quad \rho\varepsilon \quad Y_M, \tag{8}$$

and:

$$\rho\frac{D\varepsilon}{Dt} = \frac{\partial}{\partial x_i}\left[\alpha_\varepsilon \mu_{\text{eff}} \frac{\partial \varepsilon}{\partial x_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}(G_k + C_{3\varepsilon}G_b)$$
$$C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \quad R. \tag{9}$$

The terms G_k, G_b and Y_M are the same as in the standard $k \quad \varepsilon$ model. The quantities α_k and α_ε are the inverse effective Prandtl numbers for k and ε, respectively. The model constants are:

$$C_{1\varepsilon} = 1.42, \quad C_{2\varepsilon} = 1.68.$$

The scale elimination procedure in the RNG theory results in a differential equation for turbulent viscosity:

$$d\left(\frac{\rho^2 k}{\sqrt{\varepsilon\mu}}\right) = 1.72\frac{\hat{\nu}}{\sqrt{\hat{\nu}3 \quad 1 + C_v}}d\hat{\nu}, \tag{10}$$

where:

$$\hat{\nu} = \frac{\mu_{\text{eff}}}{\mu}, \quad C_v \approx 100. \tag{11}$$

The above ordinary differential equation is integrated to obtain an accurate description of how the effective turbulent transport varies with the effective Reynolds number (or eddy scale), allowing the model to correctly handle the low Reynolds number and near wall flows. In the high-Reynolds-number limit, this equation gives:

$$\mu_t = \rho C\mu\frac{k^2}{\varepsilon}, \tag{12}$$

with $C_\mu = 0.0845$, derived using the RNG theory.

The term R in the ε transport (Equation 9) that did not appear in the standard $k \quad \varepsilon$ model, is given by:

$$R = \frac{C_\mu \rho \eta^3 (1 \quad \eta/\eta_0)}{1 + \beta\eta^3}\frac{\varepsilon^2}{k}, \tag{13}$$

where:

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

$$S = \left[\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right]^{\frac{1}{2}}. \tag{14}$$

The effects of this term in the RNG ε equation can be seen more clearly by rearranging the ε transport equation in transport equations for the RNG $k \quad \varepsilon$ model.

As a result, in rapidly strained flows, the RNG model yields a lower turbulent viscosity than the standard $k \quad \varepsilon$ model. Thus, the RNG model is more responsive to the effects of rapid strain and streamline curvature than the standard $k \quad \varepsilon$ model, which explains the superior performance of the RNG model for certain classes of flow [9]. In the present work, both models are used and are compared to each other.

Boundary Conditions

Used boundary conditions are no-slip for all walls and a uniform velocity profile in the inlet for all cases. Turbulence boundary conditions are the same as Stamou et al. [11]. For k and ε, the inlet flow is assumed fully turbulent, such that characteristic values for fully turbulent channel flow are taken. Thus, $k = 3.3u*^2$ and $\varepsilon = u*^2 u_{\text{in}} h_i$ imposed at the inlet, where the shear velocity $u* = \sqrt{\tau_w/\rho}$ is established from the standard friction factors of turbulent channel flow and h_i is the inlet height of flow [11]. At the outlet, a fully developed boundary condition is made. Then, the streamwise gradients of all variables are set to zero. It is expected that the modeling of the outlet will only have a local effect on the flow field [12]. At the free surface, the rigid-lid approximation is made and the effects of free surface curvature are also neglected. Then, the symmetry condition is applied, which includes zero gradients and zero fluxes perpendicular to the boundary [12,13]. At the rigid walls, a wall function approach is used, which basically relates the wall parallel-velocity, k, and ε at the first grid point to the wall shear stress.

Solution Procedure

The governing equations are solved by a finite-volume method using boundary fitted coordinates. The momentum, turbulent kinetic energy and dissipation equations are solved for the velocity components u, v,

k and ε in the fixed Cartesian directions on a non-staggered grid. The equations are solved using the Calk-Code [14]. In this code a collocated grid is used. The velocity components at the control volume faces are computed by the Rhie-Chow [15] interpolation method and the pressure-velocity coupling is handled by the SIMPLEC algorithm. The convective terms are discretized using the QUICK scheme. TDMA-based algorithms are applied for solving the algebraic equations. The solution procedure is iterative and the computations are terminated when the sum of absolute residuals normalized by the inflow fluxes is below $10^{\ 4}$ for all the variables. The mesh points are chosen as uniform in the flow direction, but, in the normal direction, the grid points are distributed in a non-uniform manner with a higher concentration of grids close to the bed surface. Each control volume contains one node at its center, but the boundary adjacent volumes contain two nodes. Grid independency is also examined for each model.

DISCUSSION

In the first part, a simplified rectangular model is used for the modeling of the tank (Figure 1a). It is assumed that flow enters uniformly from the inlet, which is under the entrance baffle. All calculations are done in the Reynolds number (based on $H = 2.7$ m and average velocity) of 62000. A grid study was done using different grids and, finally a 60×180 grid was used for the modeling. It should be considered that in a real case, the baffle is not stretched to the free surface; but here, for simplicity, such an assumption was made, which was also done by Stamou [11]. Hence, the experimental results [11] can be used only qualitatively not quantitatively. The u-velocity profiles in some positions are shown and are compared with experimental and Stamou's numerical results (Figures 2 to 4). Velocity profiles obtained from both turbulence models

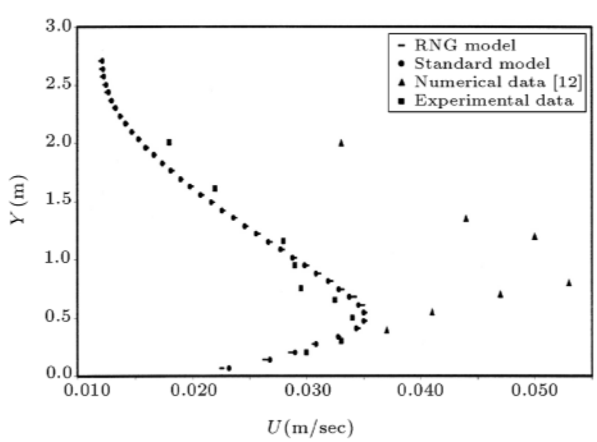

Figure 2. Velocity distribution at $x = 15$ (m) for simple case.

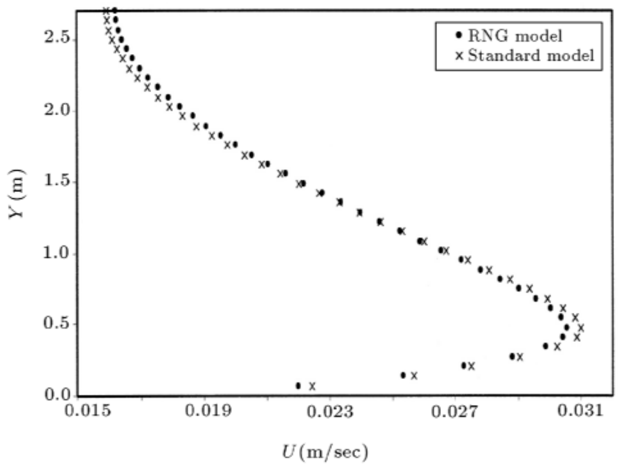

Figure 3. Velocity distribution at $x = 20$ (m) for simple case.

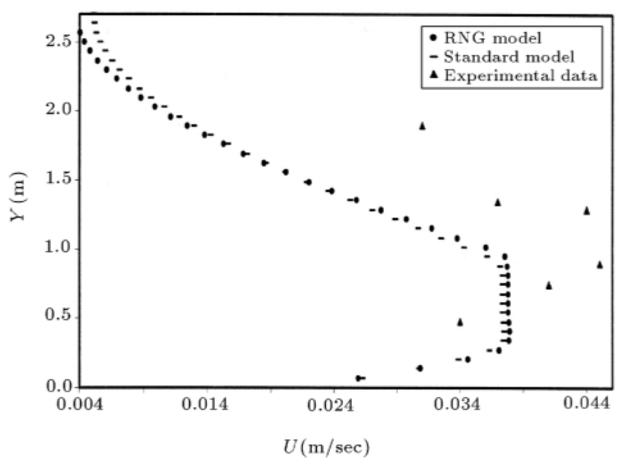

Figure 4. Velocity distribution at $x = 9$ (m) for simple case.

used in this work showed good agreement with each other, which means that in primary rectangular settling tanks, the standard k ε model is an acceptable model and will give good results. As mentioned before, experimental results cannot be discussed quantitatively and numerical results differed quantitatively from it. It was also discussed by Stamou [11]. It should be noted that the standard k ε is a high Reynolds number turbulence model, but, in circular tanks and final tanks, where density currents are present and where the concentration equation should be solved, it must have more influence. In fact, flow field details affect concentration directly. As shown in Figure 5, turbulence viscosity contours are different in two models. Turbulence viscosity contours show that a constant turbulence viscosity model is not acceptable. The circulation zone length in the simplified geometry with Re = 62000 was predicted to be about 8 (m), which had formerly been predicted by Stamou [11] to be about 6.24 (m), by Imam [10] to be about 7.4 (m)

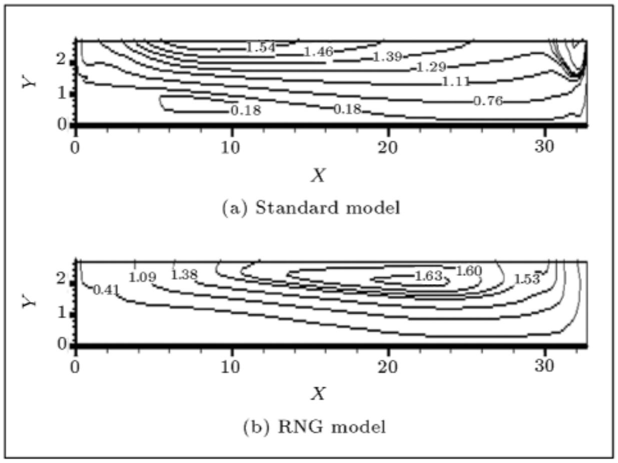

Figure 5. Turbulent viscosity contours with different turbulent models (Re = 62000) in simple geometry.

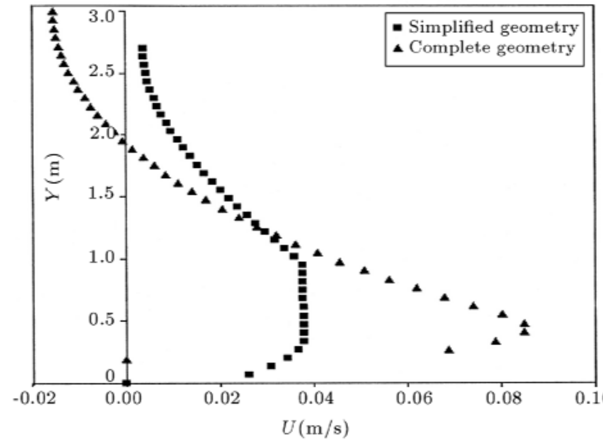

Figure 7. Comparison of velocity profiles in different geometries at distance of $x = 9$ (m) from inlet baffle.

and by experiment to be 6-9 (m).

In the second part, the inlet region and the bottom slope are also added to the geometry. It can be used for considering the effects of the inlet baffle and the simplification of geometry on the results. Inlet baffle may create a circulation region behind it, but it may spoil a short-circuiting problem. Streamlines and some velocity profiles are shown and compared with the results of the simplified geometry (Figures 6 and 7). It can be seen that simplification of the geometry caused an underprediction of the circulation region length of about 1(m).

In the last part, the effects of the existence of another baffle and its position were studied. Another baffle with a height of 1.4 (m) was added at different positions in the simplified geometry and its effect on the circulation region length was studied. In this part, only the RNG-k ε model was used. All boundary conditions are the same as in the first part. Streamlines are shown in Figure 8. Results showed that it did not affect the main circulation zone much but it created another circulation region. It also increased a probability of short-circuiting, which would lower the tanks sedimentation performance. So, a baffle, in this case, is not suitable, but it seems that, for final settling tanks, which have a higher concentration and have density currents at the tank bed, the existence of a baffle is suitable.

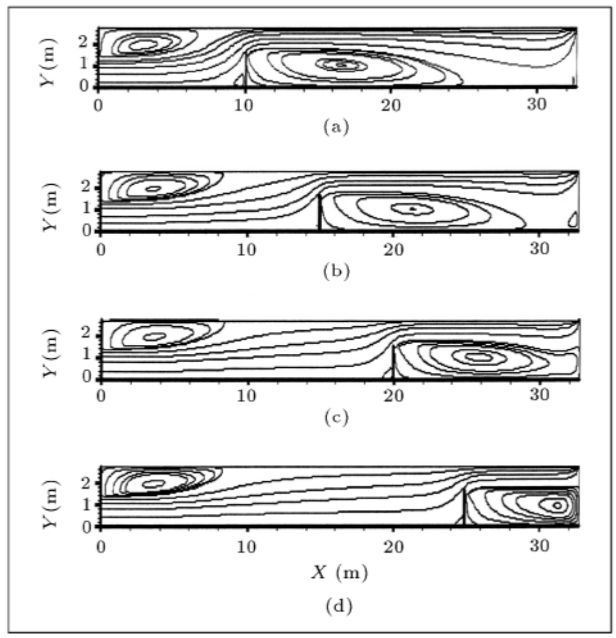

Figure 8. Streamlines in different interior baffle position.

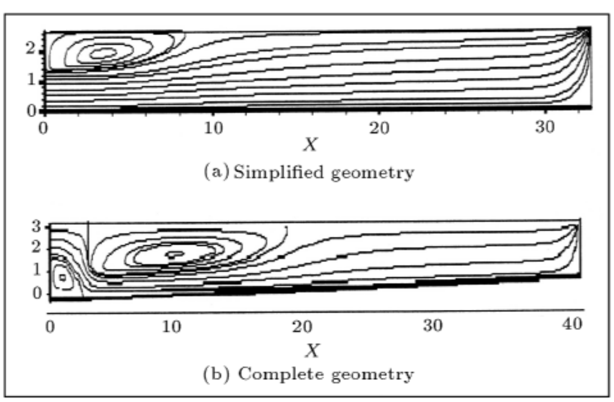

Figure 6. The influence of geometry simplification on streamlines.

CONCLUSION

Two different types of k ε (two-equation) turbulence models were studied in a settling tank of the city of Sarnia, Ontario, Canada. The results of hydrodynamic studies show that, in spite of equal velocity profiles, the RNG model prediction of the flow pattern is different. This difference could cause a better prediction of concentration, if these results were used. It must be noted that in most of the real settling tanks, large regions with circulation exist. As mentioned above, the RNG model is highly suitable for prediction of the size of the recirculation regions. So, the use of the RNG model is recommended. Another important aspect is when particle dispersion, flocculation or resuspension are considered, turbulence parameters, such as viscosity, intensity and so on, are important.

The effects of the inlet baffle were studied, which lowered the probability of short-circuiting but created a larger circulation region. Finally, in this primary settling tank, the effects of another baffle were studied which created two large circulation regions and increased the probability of short-circuiting. So, for primary settling tanks, the interior baffle is not suitable, but in tanks with very large circulation zones, especially in the final settling tanks, it seems to increase tank performance.

NOMENCLATURE

$C_{1\varepsilon}, C_{2\varepsilon}$	constants of the k ε model
C_μ, C_2	constants of the k ε model
h	average depth of settling tank
h_i	inlet depth
k	turbulent kinetic energy per unit mass
l	length of the tank
p	pressure
Re	Reynolds number $= u_i h_i/\nu$
u_i	inlet velocity
u	x velocity
v	y velocity
x, y	streamwise and normal directions

Greek Symbols

$\alpha_k, \alpha_\varepsilon$	the inverse effective Prandtl numbers for k and ε
ε	dissipation of turbulence energy per unit mass
μ	dynamic viscosity
μ_t, μ_{eff}	turbulent viscosity
ν	kinematic viscosity $= \mu/\rho$
ρ	density
$\sigma_k, \sigma_\varepsilon$	constant of the k ε model
τ_w	wall shear stress

REFERENCES

1. Dobbins, W.E. "Effects of turbulence on sedimentation", *Transactions of ASCE*, **109**(2218), pp 629-656 (1944).
2. Camp, T.R. "Sedimentation and the design of settling tanks", *Transactions of ASCE*, **3**, pp 895-936 (1946).
3. Larsen, P. "On the hydraulics of settling basins", Rep. No. 1001, Dept. of Water Resour. Engrg. Lund Inst. of Technol. Lund, Sweden (1977).
4. Schamber, D.R. and Larock, B.E. "Numerical analysis of flow in sedimentation basins", *Proc. ASCE*, **107**(HY5), pp 575-591 (1981).
5. Imam, E. and McCorquodale, J.A. "Numerical modeling of sedimentation tanks", *Proc. ASCE*, **109**(HY12), pp 1740-1754 (1983).
6. Abdel-Gawad, S.M. and McCorquodale, J.A. "Numerical simulation of rectangular settling tanks", *J. Hydraulic Research*, **23**(2), pp 85-100 (1985).
7. Celik, I., Rodi, W. and Stamou, A.I. "Prediction of hydrodynamic characteristics of rectangular settling tanks", *Int. Symposium of Refined Flow Modeling and Turbulence Measurements*, Iowa, USA (1985).
8. Adams, E.W. and Rodi, W. "Modeling flow and mixing in sedimentation tanks", *J. of Hydraulic Engineering*, **116**(7), pp 895-913 (1990).
9. Rodi, W.,*Turbulence Models and Their Application in Hydraulics*, IAHR, Delft, The Netherlands (1993).
10. Imam, E.H., *Numerical Modeling of Rectangular Clarifiers*, PhD. Thesis, University of Windsor, Ontario, Canada (1981).
11. Stamou, A.L., Adams, E.W. and Rodi, W. "Numerical modeling of flow and settling in primary rectangular clarifiers", *J. of Hydraulic Research*, **27**, pp 665-682 (1990).
12. Lyn, D.A., Stamou, A.I. and Rodi, W. "Density currents and shear-induced flocculation in sedimentation tanks", *J. of Hydraulics Eng., ASCE*, **118**(6), pp 849-867 (1992).
13. Guetter, A.K. and Jain, S.C. "Analytical solution for density currents in settling basins", *J. of Hydraulics Eng., ASCE*, **117**(3), pp 324-345 (1991).
14. Davidson, L. and Farhanieh, B., *A Finite Volume Code Employing Collocated Variable Arrangement and Cartesian Velocity Components for Computation of Fluid Flow and Heat Transfer Complex Three-Dimensional Geometry*, Chalmers University of Technology, Goteborg, Sweden (1991).
15. Rhie, C.M. and Chaw, W.L. "Numerical study of the turbulent flow past an airfoil with trailing edge separation", *AIAA J.*, **21**, pp 1525-1532 (1983).