European Water 58: 61-67, 2017.

CFD modelling of wind effect on rectangular settling tanks of water treatment plants

A. Gkesouli* and A. Stamou
Laboratory of Applied Hydraulics, School of Civil Engineering, National Technical University of Athens, 5 Heroon Polytechniou Str., 15780 Athens, Greece
* *e-mail: anthi.gkesouli@gmail.com*

Abstract: Sedimentation is a simple physical process which is widely applied on conventional water treatment plants; however, the tanks' efficiency depends on numerous factors including the wind action. In the present study, calculations are performed with a VOF model and one-phase model, which uses as an input the location of free surface and the surface velocity which are determined by the VOF model, to investigate the effect of a strong co-current wind (11 m/s) on a rectangular settling tank of a conventional water treatment plant. The results show that: (1) The flow field is highly affected by wind action, i.e. under windy conditions the tank is covered by an extended wind-generated eddy, while the suspended solids' concentration fields are influenced by the corresponding flow fields. (2) Without wind, the tank's removal efficiency is calculated equal to 85.4%, by the VOF model, and equal to 85.3%, by the one-phase model; these values are in satisfactory agreement with the experimentally determined value (86.0±1%). (3) For a co-current wind of 11 m/s, the tank's efficiency decreases to 83.4% (VOF) and 83.6% (one-phase model), which is also in satisfactory agreement with the experimentally determined value (84.0%); however, the wind effect is slight (≈2%). (4) Both the VOF and the one–phase model can predict satisfactorily the tank's removal efficiency; so the one-phase model can be used to assess wind effect on all types of settling tanks, according to the proposed methodology, since it is a significantly less time consuming model compared to the VOF model.

Key words: wind effect; settling tank; tank's performance; water treatment plant; CFD

1. INTRODUCTION

Conventional water treatment plants are still widely used for the production of potable water. The first stage of treatment in these units is performed in settling tanks where the "heavy" solids are removed from the flow, through the hoppers at the tanks' bottom, after undergoing the effect of sedimentation, also known as settling by gravity. Although the settling is a simple physical process, the efficiency of settling tanks depends on numerous factors since settling can be affected by both the complex hydraulics of the tanks and the characteristics of the solids. Wind action is one of the factors which are considered to affect the tank's efficiency (Asgharzadeh *et al.*, 2012); however, the studies that can be found in the literature regarding the wind effect are limited (Gkesouli *et al.*, 2016; Stamou and Gkesouli, 2015; Khezri *et al.*, 2012; Sivakumar and Lowe, 1990).

In the present study we perform calculations using two different approaches to investigate the effect of a strong co-current wind on the flow field and the removal efficiency of a rectangular settling tank of a conventional Water Treatment Plant (WTP), i.e. a tank with low inlet concentrations of suspended solids. Firstly, we formulate and apply a 2D two–phase CFD model (VOF) and, then, we carry out calculations with a 2D one-phase CFD model which "uses" as an input (a) the location of the free surface and (b) the surface velocity which are determined by the two-phase calculations; the results of the present study are also compared with previous studies (Gkesouli *et al.*, 2016, Stamou and Gkesouli, 2015). The proposed methodology can be used to assess the wind effect on all types of settling tanks, including rectangular and circular tanks, used for water and wastewater treatment.

2. THE MATHEMATICAL MODEL

The calculations are performed with the CFD model ANSYS-CFX 16.1. In the present section the basic equations of the model are briefly presented (ANSYS-CFX, 2017).

2.1 Flow field equations and turbulence modelling

A homogenous multiphase Eulerian – Eulerian fluid approach is used to determine the location of free surface, i.e. air and water share the same quantities (e.g. velocity and turbulence) and they are separated by a distinct resolvable interface. In this approach, the continuity (Eq. 1) and the momentum equations (Eq. 2), which involve the bulk density, ρ_m, (Eq. 4) and the bulk viscosity, μ_m, (Eq. 5) of air – water mixture, are solved to determine the 2D flow field in the tank.

Bulk continuity equation

$$\frac{\partial \rho_m}{\partial t} + \frac{\partial \rho_m U_i}{\partial x_i} = 0 \tag{1}$$

Bulk momentum equations

$$\frac{\partial \rho_m U_i}{\partial t} + \frac{\partial \rho_m U_j U_i}{\partial x_j} = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left[\mu_m\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) + RTS\right] + g_i \rho_m \tag{2}$$

Reynolds Turbulent Stresses (RTS)

$$RTS = \mu_{tm}\left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right) - \frac{2}{3}\delta_{ij}\,\rho_m\,k \tag{3}$$

$$\rho_m = r_w \rho_w + r_a \rho_a \tag{4}$$

$$\mu_m = r_w \mu_w + r_a \mu_a \tag{5}$$

where t is the time, x_i is the Cartesian coordinate in the i-direction, U_i is the common velocity, P is the pressure, g_i is the gravitational acceleration and r is the volume fraction; the subscripts 'm', 'w' and 'a' denote air - water mixture, water and air, respectively.

For turbulence modeling the SST k - ω model is applied; in the k-ω transport equations (Eq. 6 and Eq. 7) of the model, k is the turbulent kinetic energy, ω is the turbulence frequency, μ_{tm} is the eddy viscosity and P_k is the production term of k.

k - ω transport equations

$$\frac{\partial(\rho_m k)}{\partial t} + \frac{\partial(\rho_m U_i k)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu_m + \sigma_k \mu_{t_m}\right)\frac{\partial k}{\partial x_i}\right] + P_k - 0.09\rho_m k\omega \tag{6}$$

$$\frac{\partial(\rho_m \omega)}{\partial t} + \frac{\partial(\rho_m U_i\,\omega)}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\mu_m + \sigma_\omega \mu_{t_m}\right)\frac{\partial \omega}{\partial x_i}\right] + \alpha\frac{\omega}{k}P_k - \beta\,\rho_m \omega^2 + (1 - F_1)2\rho_m \sigma_{\omega 2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \tag{7}$$

F_1 and F_2 are blending functions; F_1 switches between 1 near the wall and 0 outside the boundary layer. All constants are computed by a blend from the corresponding constants of k-ε and k-ω models via a linear combination e.g. $\alpha=\alpha_1 F_1+\alpha_2(1-F_1)$. The values of the constants are the following

α_1=5/9, β_1=3/40, σ_{k1}=0.85, $\sigma_{\omega 1}$=0.5, α_2=0.44, β_2=0.0828, σ_{k2}=1, $\sigma_{\omega 2}$=0.856 (Menter *et al.*, 2003).

2.2 VOF model

As aforementioned, the location of free surface is determined via an interface-capturing method i.e. the homogeneous model, also known as Volume of Fraction (VOF) model. The VOF model determines the volume fractions of air and water in the tank (Eq. 8, Eq. 9) which vary from 0 (empty cells) to 1 (fully filled cells) while a volume conservation equation is also solved (Eq. 10).

Continuity equations

$$\frac{\partial r_w \rho_w}{\partial t} + \frac{\partial r_w \rho_w U_i}{\partial x_i} = 0 \quad (8)$$

$$\frac{\partial r_a \rho_a}{\partial t} + \frac{\partial r_a \rho_a U_i}{\partial x_i} = 0 \quad (9)$$

Volume conservation equation

$$r_w + r_a = 1 \quad (10)$$

2.3 The suspended solids equation

For calculating the suspended solids' concentration field in the tank the Algebraic Slip Model is used. The 'dirty' water (dw) which enters into the settling tank from the upstream coagulation–flocculation unit consists of 'pure' water and suspended solids of varying diameters that can be classified based on their characteristic diameter. For each class (c_i) with mass fraction, Y_{ci}, the concentration field in the tank is calculated by a mass balance equation (Eq. 11), where the slip velocity (Uc_i) is equal to the settling velocity of the class. More details can be found in Stamou and Gkesouli (2015).

$$\frac{\partial(\rho_{dw} Y_{c_i})}{\partial t} + \frac{\partial\left(\rho_{dw} Y_{c_i}\left(U_j + U_{c_i j}\right)\right)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(\mu_{dw}\frac{\partial Y_{c_i}}{\partial x_j} + \frac{\mu_{t\,dw}}{0.9}\frac{\partial Y_{c_i}}{\partial x_j}\right) \quad (11)$$

Since the 'dirty' water is a mixture of 'pure' water and suspended solids, its density, viscosity and eddy viscosity are bulk quantities which are calculated by equations similar to Eq. 4 and Eq. 5. In the present study, all quantities of water included in Eq. 1 to Eq. 10 refer to dirty water.

3. TANK'S GEOMETRY AND CALCULATION DETAILS

3.1 Geometry of the settling tank

The settling tank is rectangular with length equal to 73.2 m, width equal to 14.4 m and average water depth equal to 3.5 m. The 'dirty' water enters into the tank via 4 rectangular openings (1.5 m x 0.35 m), which are placed at y=0.55 m above the tank's bottom, and overflows via 3 outlet channels, with length equal to 10 m, which are installed at x=63.2 m. The sludge, which is collected at the bottom of the tank, is removed every day via 2 hoppers. In the present study the tank is modeled 2 dimensionally considering that the 'dirty' water enters into the tank via a slot opening equal to 0.15 m and overflows via a rectangular outlet weir which is placed at x=63.2 m, i.e. at the beginning of the outlet channels.

3.2 Field measurements

Hourly measurements of wind direction and wind velocity near the tank were performed with an anemometer for a total period of 2 years; these measurements showed that the strongest co-current winds in the study area have velocities equal to W≈11.0 m/s (at y≈10 m above the water surface). During these "strong wind periods" turbidity at the tank's inlet and outlet was measured by on-line turbidimeters while the tank's inlet flow rate was also recorded. The removal efficiency of the tank during the strongest recorded co-current wind (W≈11 m/s) was estimated equal to ≈84.0%, for steady state flow conditions, based on the above mentioned measurements.

3.3 Methodology of calculations

Firstly, we performed two-phase calculations with the VOF model in order to determine the flow field, the suspended solids' concentration field and the removal efficiency of the tank and, consequently, to assess the wind effect on tank's performance; based on these calculations we also derived (a) the location of free surface and (b) the surface velocity caused by wind action. Then, we used these data as an 'input' in a significantly less time consuming one-phase model and we performed the same kind of calculations (velocity and solids' concentration field, removal efficiency); the calculated removal efficiencies by the VOF model and the one-phase model were then compared. The proposed methodology can be used to assess the wind effect on all types of settling tanks.

3.4 Computational domains

In the case of two-phase calculations, the computational domain had length equal to 63.2 m, i.e. equal to the tank's length, and height equal to 13.5 m, i.e. equal to the tank's expected average water depth plus 10 m (representing the air domain). At the borders of the domain the following boundary conditions were applied: (i) inlet of the tank: horizontal velocity equal to 0.115 m/s deduced by the measured inlet flow rate of the tank (0.25 m^3/s) divided by the inlet area, (ii) outlet of the tank: pressure and direction of the flow, (iii) walls and bottom of the tank: no slip walls, (iv) side walls: symmetry, (v) inlet of air domain: log law theoretical wind velocity profile and (vi) outlet of air domain: pressure and direction of the flow. The location of free surface was calculated by the VOF model.

In the case of one-phase calculations, the computational domain consisted exclusively of the tank, i.e. we did not take into account the air domain; its length was equal to 63.2 m and the water depth was equal to ≈3.46 m (determined by the VOF model). At the free surface we applied the wind-caused surface velocity which was also calculated by the VOF model; the rest of the boundary conditions were the same as in two-phase calculations.

Both grids were selected after a series of grid independency tests and they were structured and refined at tank's inlet, outlet and bottom and near the free surface.

3.5 Scenarios of calculations

In order to investigate the wind effect on tank's performance and removal efficiency, we carried out calculations (a) for no wind conditions (W=0 m/s) and (b) for the strongest co-current wind which was recorded at the area of study (W=11 m/s). It is noted that the number of the classes at the tank's inlet, their inlet concentrations and their settling velocities were taken from a previous study (Gkesouli *et al*., 2016).

4. RESULTS AND DISCUSSION

In Figure 1 the calculated flow field in the tank is presented for (a) W=0 m/s, based on the one-phase calculations, (b) W=11 m/s, based on the VOF model, and (c) W=11 m/s, based on the one-phase calculations using the surface velocity predicted by VOF. The corresponding calculated suspended solids' concentration fields are presented in Figure 2. The tank's calculated removal efficiencies by the VOF and one-phase model are shown in Table 1 together with the experimentally determined values and the removal efficiencies calculated by a 3D model and a 2D model applied in previous studies.

Figure 1 shows that under no wind conditions the inlet jet creates an anticlockwise eddy that covers the first ≈20 m of the tank while at the rest of the tank the water flow is practically parallel to axis x. The presence of the strong co-current wind 'suppresses' the anticlockwise eddy to approximately 12 m and at the rest of the tank an extended clockwise eddy is created due to wind action; the structure of the flow field predicted by the VOF and one-phase model is practically the same while VOF predicts relatively higher velocities. Therefore, it is obvious that the presence of wind highly affects the hydraulic regime in the tank since it leads to the creation of a massive dead zone. Figure 2 depicts that the suspended solids' concentration fields are highly affected by the respective flow fields e.g. for W=0 m/s a 'sludge' layer is created in the region of the anticlockwise eddy while for W=11 m/s the suspended solids' concentration field in the wind-generated eddy is more uniform. As expected in the case of the co-current wind, the iso-concentration contours are shifted upwards, towards the free surface, after undergoing the effect of the wind–generated eddy; thus, the outlet concentration increases and the efficiency of the tank is reduced. The suspended solids' concentration fields predicted by the VOF (Fig. 2b) and one-phase model (Fig. 2c) are similar.

Table 1 shows that without wind the tank's removal efficiency is calculated equal to 85.4%, by the VOF model, and equal to 85.3%, by the one-phase model; these values are practically equal and are also in satisfactory agreement with (a) the experimentally determined value (86.0±1%) and (b) the values determined by a 3D model (85.7%) and a different 2D model (83.1%). For W=11 m/s, the calculated removal efficiencies decrease to 83.4% (VOF model) and 83.6% (one-phase model). It is obvious that the wind has a detrimental effect on tank's settling performance; however, this reduction is low (≈2%). These values are also in satisfactory agreement with previously performed calculations with a 3D model (84.2%) and a different 2D model (82.4%) and with the experimental value (84.0%) which was determined using the turbidity measurements at the tank's inlet and outlet during a strong wind event in the tank in May 2015. These results show that both the VOF model and the one-phase model can satisfactorily predict the removal efficiency of the tank; however, it is noticed that the VOF model is significantly more time consuming compared to the one-phase model. For the performance of the present calculations we used an Intel Core i7 - 3.33 GHz desktop and the CPU time/Real time was equal to 0.007 in the case of the one-phase model and equal to 1.8 in the case of the VOF model. Therefore, the proposed methodology (see also section 3.3) can be applied to assess the wind effect on tank's efficiency and decrease significantly the required computational time.

Table 1. Removal efficiencies (%) of the tank

Wind velocity	**W=0 m/s**	**W=11 m/s**
Experimentally determined value	**86.0 ± 1.0[b]**	**84.0**
2D model - VOF model	85.4	83.4
2D model - One-phase model / Surface velocity	85.3	83.6
3D model	85.7[b]	84.2[a]
2D model - FLOW 3D	83.1[b]	82.4[b]

[a] *Obtained from Stamou and Gkesouli (2015)*
[b] *Obtained from Gkesouli et al. (2016)*

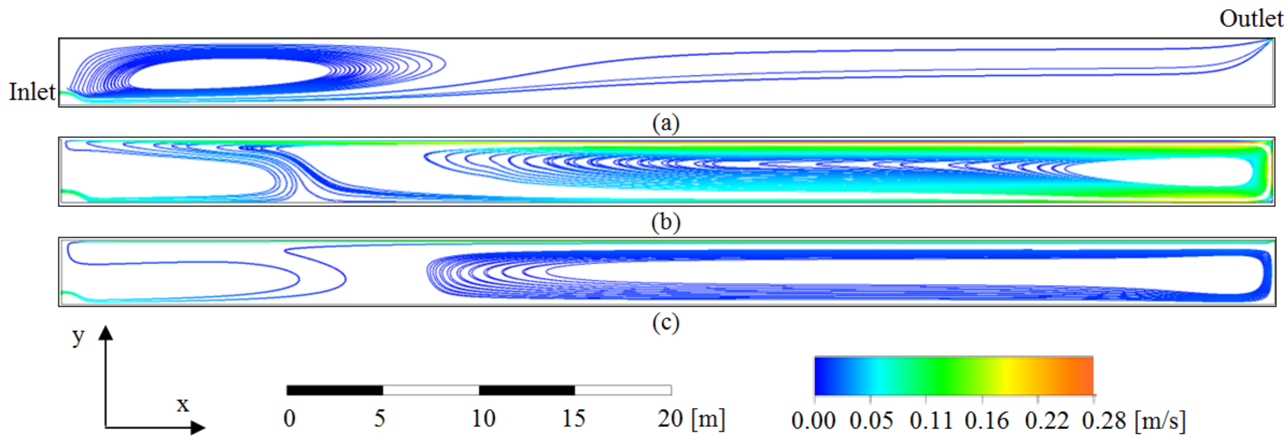

Figure 1. Calculated flow field for (a) W=0 m/s, (b) W=11 m/s (VOF model) and (c) W=11 m/s (one-phase model).

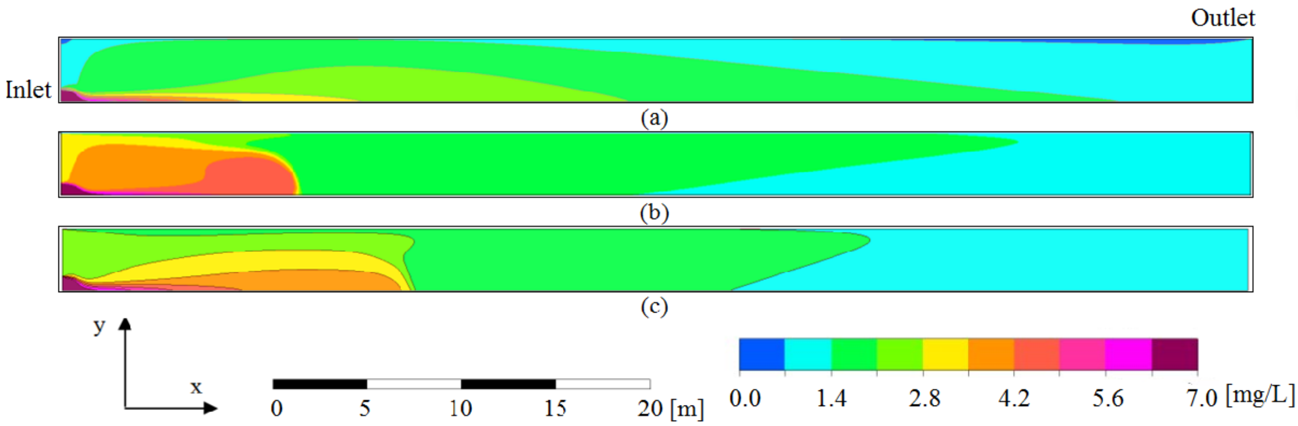

Figure 2. Suspended solids' concentration field for (a) W=0 m/s, (b) W=11 m/s (VOF model) and (c) W=11 m/s (one-phase model).

5. CONCLUSIONS

Calculations were performed to assess the wind effect on a rectangular settling tank of a conventional WTP with two approaches: we performed (a) two-phase calculations with a VOF model and (b) calculations with one-phase model in which we used as an input the location of free surface and the surface velocity which were determined by the VOF model. The results of these calculations showed that: (1) The flow field in the tank is highly affected by the wind action; under windy conditions the tank is covered by an extended wind-generated eddy. The suspended solids' concentration fields are influenced by the corresponding flow fields. (2) The tank's removal efficiency without wind is calculated equal to 85.4%, by the VOF model, and equal to 85.3%, by the one-phase model; these values are in satisfactory agreement with the experimentally determined value (86.0±1%) and previous studies (85.7% - VOF, 83.1% - one phase model). (3) Under windy conditions (W=11 m/s), the tank's efficiency decreases (83.4% - VOF, 83.6% - one-phase model); however, the effect is slight (≈2%). (4) The removal efficiencies predicted by the VOF and the one –phase model are practically the same; so the proposed methodology i.e. performing calculations with one-phase model which uses as an input the location of free surface and the surface velocity predicted by the VOF model can reduce significantly the required computational time. The proposed methodology can be used to assess the wind effect on all types of settling tanks.

REFERENCES

ANSYS-CFX, Release 16.1. Available at: http://www.ANSYS.com (accessed 14/02/2017).

Asgharzadeh, H., Firoozabadi, B. and Afshin, H. 2012. Experimental and numerical simulation of the effect of particles on flow structures in secondary sedimentation tanks. Journal of Applied Fluid Mechanics 5 (2), 15–23.

Gkesouli, A., Nitsa, M., Stamou, A., Rutschmann, P. and Bui, M. D. 2016. Modeling the effect of wind in rectangular settling tanks for water supply. Desalination and Water Treatment 57 (54), 26345-26354.

Khezri, S. M., Biati, A. and Erfani, Z. 2012. Determination of the effect of wind velocity and direction changes on turbidity removal in rectangular sedimentation tanks. Water Science and Technology 66 (12), 2814–2820.

Menter, F. R., Kuntz, M. and Langtry, R. 2003. Ten Years of Industrial Experience with the SST Turbulence Model. Turbulence, Heat and Mass Transfer 4, 625-632.

Sivakumar, M. and Lowe, S. A. 1990. Simulation of the effect of wind on rectangular sedimentation tanks. In: Conference on Hydraulics in Civil Engineering, Sydney, Australia, 74–78.

Stamou, A. and Gkesouli, A. 2015. Modeling settling tanks for water treatment using computational fluid dynamics. Journal of Hydroinformatics 17 (5), 745-762.