A large eddy simulation study to assess low-speed wind and baffle orientation effects in a water treatment sedimentation basin
Danial Goodarzi, Kaveh Sookhak Lari and Abolghasem Alighardashi

ABSTRACT

Hydraulic performance of clarifiers in water and wastewater treatment plants significantly affects the settling efficiency of suspended particles. Structural and ambient parameters can deteriorate this performance. Through a verified three dimensional numerical study, we evaluated hydraulic performance and settling efficiency in a rectangular clarifier with a nominal hydraulic retention time (HRT) of 1 h and options for structural baffles with angles of 20°, 30°, 45° and 70°. Large eddy simulation and Lagrangian particle tracing were used to trace particles 80 to 850 μm in diameter. A passive scalar tracer study was conducted to reveal discrepancies in nominal and real HRT. By posing a 5 m/s wind, ten different scenarios were simulated. The wind caused 17% and 6% reduction in HRT and settling efficiency, respectively. Baffles improved these indicators with the 45° baffle showing the best performance with an approximate settling efficiency of 93%. The study highlighted the importance of using baffles, in particular for small size particles for which influencing factors such as wind deteriorate their settling efficiency.

Key words | baffle, large eddy simulations, particle settling, sedimentation basin, wind effect

INTRODUCTION

Treated water quality should meet strict environmental and public health standards. Various physical, chemical and biological processes are applied for water and wastewater treatment. Among these, the primary physical process is to remove suspended particles from the water stream through ordinary or enhanced gravitational separation. Depending on the quality required and arrangement of other processes, this is mostly performed in clarifiers of various layout and size (Tebbutt 1998). Successful gravitational separation is an affordable technique to also reduce chemical and biological loadings in the treated water (Sookhak Lari & Kargar 2015). However, as the specific density of common suspended particles is not significantly greater than one and also their diameter is within a range of tens to hundreds of μm, any disruption and turbulence in the flow pattern can deteriorate the settling efficiency of the particles (Khezri et al. 2012). This may be due to thermal stratification, wind effects and poor hydraulic design. Such issues can induce turbulence in a clarifier which, in turn, causes re-suspension of the settled particles (Shiono & Teixeira 2000; Khezri et al. 2012). Hence, performance of a sedimentation basin relies on its flow characteristics.

Field-scale observations, experimental pilot studies and numerical simulation have been used to study performance of clarifiers. As examples of the first two, seasonal field observations were conducted in a water treatment plant to study enhanced gravitational separation in circular clarifiers (Sookhak Lari & Kargar 2015). Flow velocity measurements were applied in a 4:1 pilot study to characterize turbulence in a baffled contact tank (Shiono & Teixeira 2000). Khezri et al. (2012) evaluated wind impact on a pilot scale sedimentation tank and found that the efficiency decreased from 61.24% to 46.04% in the presence of wind. As examples for computational simulation, a numerical study on clarifier inlet configuration showed that baffles reduce water recirculation zone and enhance the settling performance (Goula et al. 2008). Asgharzadeh et al. (2012)
showed that sedimentation could be affected by wind effects. Stamou & Gkesouli (2015) found that by applying baffles inside a specific tank, settling efficiency would increase from 68.07% to 71.04% in a windy condition. Baffle position and configuration was studied by Tamayol et al. (2008) who showed that installation of baffles in recirculation regions would decrease formation of dead zones. Al-Sammarraee & Chan (2009) and Al-Sammarraee et al. (2009) showed that by adding three 20° baffles to a sedimentation tank, removal efficiency would improve. Recently, application of baffles in a windy condition was studied by Gkesouli et al. (2016). It was shown that in windy conditions, adding one or two baffles would improve settling efficiency.

Large eddy simulation (LES) mimics the 3D nature of turbulent fluctuations. It is more accurate than Reynolds-averaged turbulence models (Sookhak Lari et al. 2011) and computationally affordable in comparison with direct numerical simulation (Al-Sammarraee & Chan 2009). LES has successfully been used to study baffles performance in rectangular clarifiers (Al-Sammarraee & Chan 2009).

Rectangular clarifiers are easier to construct in comparison with circular clarifiers. The possibility of using mutual walls between two adjacent tanks make these basins a favorable choice where the resources and expertise are scarce (Sookhak Lari & Kargar 2013). Here and for the first time, we aim to use LES to investigate simultaneous effects of baffle orientation and low-speed wind on settling efficiency of a range of particle sizes and hydraulic performance of a rectangular clarifier.

### METHODS

Despite their several successful performances (Zhang et al. 2015), Reynolds-averaged modelling approaches like k-ε have some limitations in studying zones of intense shear and flow with low or moderate Reynolds numbers (Murakami 1993; Rodi 1997). We then use LES to better resolve the turbulence and also to follow the same approach in Al-Sammarraee & Chan (2009).

LES simulates larger fluctuations in flow field parameters (e.g. instantaneous velocity and pressure) while smaller fluctuations are filtered out. The LES filtered momentum equation (to be considered along with the filtered continuity equation) is (Pope 2000, p. 582):

\[
\frac{D U_j}{D t} = \nu \frac{\partial^2 U_j}{\partial x_i \partial x_i} - \frac{1}{\rho} \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}^f}{\partial x_j},
\]

(1)

where the substantial derivative (based on the filtered velocity vector \( \bar{U} \)) is

\[
\frac{D}{D t} \equiv \frac{\partial}{\partial t} + \bar{U} \cdot \nabla
\]

(2)

where \( U_j \) represents filtered velocity of the \( j \)-th direction \([\text{LT}^{-1}]\), \( x_i \) is the \( i \)-th coordinate, \( t \) and \( \rho \) are time \([\text{T}]\) and fluid density \([\text{ML}^{-3}]\), \( \bar{p} \) is the filtered pressure \([\text{ML}^{-1} \text{T}^{-2}]\) and kinematic viscosity of the fluid is represented by \( \nu \) \([\text{ML}^{-1} \text{T}^{-1}]\) (Pope 2000, p. 582). With \( \tau_{ij}^f \) (anisotropic residual stress tensor) given by a residual stress model (to close Leonard, cross and SGC Reynolds stresses), it is possible to determine filtered velocity and pressure fields. The simplest approach is the Smagorinsky’s model (Pope 2000, p. 587):

\[
\tau_{ij}^f = -2(c_s \Delta)^2 S \bar{S}_{ij}
\]

(3)

where \( S \) is the characteristics filtered rate of strain

\[
S = \left( 2 \bar{S}_{ij} \bar{S}_{ij} \right)^{1/2}
\]

(4)

Also, \((c_s \Delta)^2 S \) is known as the eddy viscosity. Indeed, Smagorinsky’s model includes a linear eddy-viscosity model to relate \( \tau_{ij}^f \) to the filtered rate of strain (\( \bar{S}_{ij} \)). The Smagorinsky constant is \( c_s \sim 0.17 \) and \( \Delta \) is the filter width. The value of \( c_s \) has shown to be different in different flow regimes. Dynamic models can determine appropriate local values for \( c_s \) (Pope 2000, p. 619), in particular for the sensitive near-wall and interfacial treatments of high Schmidt number scalars transport (Sookhak Lari et al. 2015). However, application of Smagorinsky’s model in this study is justified as we are more interested in the bulk flow (rather than the viscous sublayer). Also, no multi-phase flow or wall reaction is included. Therefore, a van Driest damping function (to the length scale) is used in Smagorinsky’s model to vanish the eddy viscosity close to the solid boundaries (Pope 2000, p. 599). This has been shown to produce accurate results with respect to the velocity field (Bird et al. 2007, p. 164).

### The domain and flow characteristics

Figure 1 depicts the longitudinal sedimentation tank used in this study. The configuration of the tank was adopted from Al-Sammarraee & Chan (2009). The tank is 175 m³ in volume, 20 m in length, 3 m in width and 2–4 m in depth. The outlet weir is 5 m long and 0.6 m wide. The tank also has three vertical baffles. We considered options of 20, 30,
45 and 70 degrees as the angle $\alpha$ between the baffles and the depth-wise direction.

Table 1 introduces the particles’ characteristics and their mass fraction in the raw water. Density of particles in the raw water is 1,000 mg/L. The particles and water density are 1,066 kg·m$^{-3}$ and 998.2 kg·m$^{-3}$, respectively. Turbulent coagulation is ignored due to the very low volumetric fraction of the particles in the tank. Also, due to the very low abundance of particles in the raw water, the effect of the particles on the flow field is ignored. Therefore, we first simulate the flow field and then use the velocity field results to trace the particles (Al-Sammarraee & Chan 2009). Stochastic discrete particle procedure is used to simulate the dispersion caused by turbulence (Goula et al. 2008). Slip velocity between particles and flow was neglected (Huggins et al. 2004).

**Simulation scenarios**

Ten simulation scenarios were considered: (1) no wind no baffle (nw-nb); (2) no wind three 20° baffles (nw-b20); (3) no wind three 30° baffles (nw-b30); (4) no wind three 45° baffles (nw-b45); (5) no wind three 70° baffles (nw-b70); and, finally, scenarios 6–10 including a wind flow (w): (6) w-nb; (7) w-b20; (8) w-b30; (9) w-b45; and (10) w-b70. The wind in this study is considered as a typical wind in urban
area, 5 m/s at a height of 10 m above the basin (Keyhani et al. 2010).

ANSYS® Fluent® 16.1 was used to perform the simulations. On a 16-node cluster, the entire simulations took almost 390 h to complete. The continuity and momentum conservation equations were solved by a pressure-based coupled unsteady solution algorithm with bounded central differencing. Volume fraction for the particle concentration computed by choosing second-order upwind scheme. Domain walls were assumed to be smooth and standard (Lauder & Spalding 1974).

The automatic mesh generation formed a mesh with typical dimensions varying from $10^{-3}$ m near the surface and boundaries to a few centimeters at the center of the tank. A mesh-dependency study was also conducted. Grids of 0.5, 1.3, 1.6 and 1.7 million cells were chosen to investigate the impact of the mesh density on the simulation. The results discrepancies were insignificant. The final grid contained 0.5 million hexahedral elements which has an almost 5-times higher resolution than the mesh used by Al-Sammarraee & Chan (2009). Automatic time steps varied from $10^{-3}$ s to 0.05 s.

To simulate the particles transport, a Lagrangian approach (Discrete Phase Model in FLUENT) was used. The motion of particles is determined by solving a set of ordinary differential equations (ODEs) for the particle velocity and position, at every time step. For decades, researchers have studied various conditions and forces acting on the particles to derive such ODEs (Squires & Simonin 2006; De Marchis et al. 2016). The magnitude of these forces are functions of several parameters including abundance, diameter and density of particles. Here we assume no inter-particle collisions. Also the particle volumetric fraction in this study is much less than 12% (Al-Sammarraee & Chan 2009). The flow field is then determined first and the dominate force in the ODE governing particles acceleration is the Stokes (viscous) drag (De Marchis et al. 2016). The governing ODEs describing particle’s position in time can be found in the literature (e.g. De Marchis et al. 2016, Equations (3)–(5)).

To improve the convergence, PISO algorithm was used and the allowed residual for convergence was set to $10^{-6}$. Mass-inlet was defined for the inlet flow (influent) and pressure-outlet for the outflow (effluent). The impact of the wind was applied via substituting the equivalent shear stress on the surface of the water:

$$\tau = -\rho_a C_D U^2$$

where $\rho_a$ is the air density [$\text{ML}^{-3}$], $U$ is the wind velocity [$\text{LT}^{-1}$] and $C_D$ is drag coefficient for momentum ($\sim 10^{-3}$) (Edson et al. 2013; Bryant & Akbar 2016). At a critical speed, fetch and elapsed time, the wind can form (visible) surface waves which, in turn, increase water surface roughness and, therefore, the assumption of smooth flow at the interface (or the pure shear stress transfer) may be violated. However, the record of water surface displacement by Plate et al. (1969) shows that for $U_\infty = 5$ m/s ($U_\infty$ is the outer layer velocity of the wind) and a fetch of 5–4 m, the steady-state wavelength would be less than a millimeter and the interface may still be regarded as smooth (Sookhak Lari et al. 2013). Indeed for a winds and fetch significantly beyond the critical point, the effect of the wind and the interface deformation should be considered using more advanced techniques like two-phase flow simulation through volume of fluid method (Xie 2017).

### Residence time distribution

We also conducted a tracer study to examine the hydraulic performance of the tank (Sookhak Lari 2013). Instantaneous injection of a passive tracer control mass at the inlet and measuring the outlet concentration produced residence time distribution functions (RTD) for all the scenarios.

The solute mass transport equation is (Huggins et al. 2004).

$$\frac{\partial c}{\partial t} + U. \nabla c = \nabla [(D + D_t)c]$$

where $c$ is the concentration of the tracer [$\text{ML}^{-3}$], $v$ is the velocity vector [$\text{LT}^{-1}$], $D$ is the molecular diffusion coefficient [$\text{L}^2\text{T}^{-1}$] and $D_T = (v_T/\Theta_T)$, where $v_T$ is the turbulent viscosity and [$\text{L}^2\text{T}^{-1}$] and $\Theta_T \approx 1$ is the turbulent Schmidt number. The real averaged hydraulic retention time (HRT) is

$$HRT_R = \frac{\int_0^\infty tc(t)\,dt}{\int_0^\infty c(t)\,dt}$$

and the variance is expressed as

$$\sigma^2 = \frac{\int_0^\infty (t-HRT_R)^2 c(t)\,dt}{\int_0^\infty c(t)\,dt}.$$
The efficiency of the tank with respect to the residence time is the ratio of the real and nominal HRT:

\[ e = \frac{\text{HRT}_R}{\text{HRT}_N} \quad (9) \]

Finally, short circuiting index is defined as:

\[ \text{Short circuiting index} = \frac{t_i}{\text{HRT}_R} \quad (10) \]

where \( t_i \) is the time at which tracer first appears.

**Verification**

We compared the settling efficiency of the particles in nw-nb case with the counterpart experimental values in Al-Sammarraee et al. (2009) for an identical tank and flow conditions. Results are depicted in Figure 2 which show acceptable agreement between the two studies. The discrepancy of the results for the range of 200–300 \( \mu \text{m} \) is greater than the discrepancy for larger and smaller particles. A possible reason may be some physical characteristics (like the real shape and electrical charge) of the particles at this range which is not reported in the original study. To further test the accuracy of our simulation framework, we also simulated the rectangular sedimentation tank reported in Liu et al. (2009). We intended to evaluate performance of the framework with respect to the mean velocities. Indeed, we investigated if the Smagorinsky’s model assumptions can have significant effects at a scale and for hydraulic characteristics of a clarifier. Figure 3 depicts the numerical simulation results versus the velocity measurements by Liu et al. (2009) (Case 3). Good agreement between the two studies is observed.

**RESULTS AND DISCUSSION**

**Flow pattern**

Figure 4 illustrates the computed RTD for the scenarios w-nb, w-b20, w-b30, w-b45 and w-b70. The tracer departing the basin through a short circuiting was identified via a primarily rising RTD followed by a long tail indicating existence of some recirculation regions (dead zones) (Sookhak Lari 2013). In general, w-b45 demonstrated a less-affected concentration plume. On the contrary, w-nb was the case showing the greatest dispersion. Results for w-b20 and w-b70 are almost identical.

Table 2 shows the hydraulic characteristics of selected scenarios. The most deviated retention time (with respect to the nominal retention time) was observed in w-nb (0.81 h) while implementing the 45° baffles improved the retention time the most (0.96 h). This also indicates that short circuiting and the existence of dead zones were more pronounced for w-nb. In general, applying baffles improved the hydraulic performance of the tank with the 45° baffles showing the greatest improvement. The average performance of the w-b70 and w-b20 are almost the same. The w-b20 case performs better with respect to the HRT, while w-b70 better diminishes the short circuiting.

Figure 2 | Comparisons of the experimental data in Al-Sammarraee et al. (2009) with the numerical results for the case nw-nb.
Particles settling efficiency

Table 3 reports the settling efficiency characteristics of all the scenarios and particle classes. As anticipated, small particles settling tendency is more affected by the wind effect and baffles configurations. Mass and inertia of particles larger than 350 μm are high enough to allow complete sedimentation. In order to calculate the removal efficiency for all the particle classes, the inlet and outlet mass flux of each particle class were measured. The removal efficiency of the tank with no baffles and in a windy condition drop to 88.79%. The best performance is also observed in the tank with 45° baffles, both in presence and absence of the wind. The wind effect is more pronounced for smaller particles for all baffle configurations. Also the 70° and 20° baffles show almost identical results with respect to the overall efficiency.

Particles with the size of 80 μm are the most affected group. To investigate the findings in Table 3 in more detail, velocity magnitudes of class 1 particles (size 80) are shown in Figure 5 for all four baffle configurations in the windy condition. The wind causes a horizontal flow on the surface of the tank. In w-b20, the downwards penetration of high-speed surface stream is greater than in the other cases. For the case of 70° baffles, it is seen that the high-speed surface current does not get mixed with
the bulk flow. This indicates formation of dead zones inside the tank and, therefore, an augmented short circuiting effect. This is consistent with the results in Table 2 in which the w-b70 case shows the worst performance with respect to the short circuiting effect (among the cases including baffles). Compared to the w-b20 case, the figure shows a more uniform condition at the bottom of the tank for the case of 45° baffles. Also, the w-b45 case shows more mixing in the bulk flow compared to the w-b70 case. These together yield the better overall performance of the w-b45 case.

Figure 6 shows the concentration of the settled particles along the length of the tank for the four baffle orientations. Analysis of the velocity vectors show that the locations at which the downwards flow directed by the baffles hits the bottom have less concentrations. These locations show some offset between different cases due to the baffles angle. Clearly, the w-b45 scenario shows a greater settled mass along the length of the tank. This is consistent with what was observed in Figure 5 and Table 3. It was observed in Figure 5 that the velocity of class 1 particles below the weirs was more uniform in w-b45 compared to the other cases (a smaller dead zone). This is also observed in Figure 6 where the concentration of particles at the end of the tank raises for the w-b45 case.

### CONCLUSIONS

We conducted series of LES to investigate settling efficiency of a typical rectangular sedimentation basin which is widely used in water and wastewater plants. The results were also verified versus existing experimental data in the literature.

For the first time, we used LES to simultaneously quantify negative impact of a low-speed surface wind on hydraulics of the tank (through a separate tracer study) and removal efficiency for a wide range of particle sizes and, meanwhile, to find out the optimum baffle orientation to diminish these adverse effects. Such simulation strategy can reveal optimum configuration of internal baffles to enhance settling characteristics of particles ranging from tens to hundreds of micrometers in diameter. In particular, the results showed that increasing the baffles’ angle from 20° to 45° or decreasing from 70° to 45° would enhance
the tank performance both with respect to the retention time and the settling efficiency.

LES by its nature is a 3D study approach and, therefore, it is possible to have better representatives short circuiting, mixing and recirculating features of the tank under various ambient and internal conditions. We showed that by using a moderate parallel processing computational facility, it would still be affordable to study these tanks through advanced computational techniques such as LES. This is, in particular, due to the hydrodynamic features of these tanks which usually comprise regions with relatively low Reynolds numbers.

Results and approaches presented here can be used to build up a study framework for similar cases which

<table>
<thead>
<tr>
<th>Table 3</th>
<th>Particle settling efficiency (%) and total settling efficiency (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Size (μm)</td>
<td>Mass fraction</td>
</tr>
<tr>
<td>80</td>
<td>0.0411</td>
</tr>
<tr>
<td>120</td>
<td>0.0696</td>
</tr>
<tr>
<td>170</td>
<td>0.1002</td>
</tr>
<tr>
<td>200</td>
<td>0.1213</td>
</tr>
<tr>
<td>250</td>
<td>0.1276</td>
</tr>
<tr>
<td>350</td>
<td>0.1308</td>
</tr>
<tr>
<td>450</td>
<td>0.1192</td>
</tr>
<tr>
<td>550</td>
<td>0.1065</td>
</tr>
<tr>
<td>650</td>
<td>0.0812</td>
</tr>
<tr>
<td>750</td>
<td>0.0601</td>
</tr>
<tr>
<td>850</td>
<td>0.0424</td>
</tr>
<tr>
<td>Total</td>
<td>1</td>
</tr>
</tbody>
</table>

Figure 5 | Particles velocity magnitudes of class 1 particles, (a) w-b20, (b) w-b30, (c) w-b45, (d) w-b70. Note that L should be identical for all baffle orientations (see Figure 1).
eventually can lead to more efficient water and wastewater treatment processes. Consequently, public and environmental risks are better assessed and managed. Study of thermal stratification and buoyancy driven flows are some possible avenues for further research.

**ACKNOWLEDGEMENTS**

We thank Dr Ka Yu Cheng (CSIRO Land and Water) for his valuable comments.

**REFERENCES**


Liu, B., Ma, J., Luo, L., Bai, Y., Wang, S. & Zhang, J. 2009 Two-dimensional LDV measurement, modeling, and optimal


