Optimization design of submerged propeller in oxidation ditch by computational fluid dynamics and comparison with experiments
Yuquan Zhang, Yuan Zheng, E. Fernandez-Rodriguez, Chunxia Yang, Yantao Zhu, Huiwen Liu and Hao Jiang

ABSTRACT
The operating condition of a submerged propeller has a significant impact on flow field and energy consumption of the oxidation ditch. An experimentally validated numerical model, based on the computational fluid dynamics (CFD) tool, is presented to optimize the operating condition by considering two important factors: flow field and energy consumption. Performance demonstration and comparison of different operating conditions were carried out in a Carrousel oxidation ditch at the Yingtang wastewater treatment plants in Anhui Province, China. By adjusting the position and rotating speed together with the number of submerged propellers, problems of sludge deposit and the low velocity in the bend could be solved in a most cost-effective way. The simulated results were acceptable compared with the experimental data and the following results were obtained. The CFD model characterized flow pattern and energy consumption in the full-scale oxidation ditch. The predicted flow field values were within $-1.28 \pm 7.14\%$ difference from the measured values. By determining three sets of propellers under the rotating speed of 6.50 rad/s with one located 5 m from the first curved wall, after numerical simulation and actual measurement, not only the least power density but also the requirement of the flow pattern could be realized.

Key words | actual measurement, computational fluid dynamics (CFD), energy consumption, flow field, submerged propeller

INTRODUCTION
The oxidation ditch, as an intermittent cycle extended aeration system, has been widely employed due to its simplicity of operation and high performance of treatment (Grady et al. 2011). In order to ensure its treatment effect and avoid sludge settling, the inside flow should be circulated around with a velocity of no less than 0.15 m/s (Wu et al. 2012). Therefore, the submerged impeller in an oxidation ditch is a major energy consumer as it provides power for the flow. However, it is still a challenge for wastewater treatment plants to operate oxidation ditch systems successfully because of technology and economy issues.

Since the treatment efficiency relies heavily on the flow field in an oxidation ditch, a good understanding of hydraulics of the ditch is a precondition for a successful design (Yang et al. 2011). However, there is a limited understanding of the hydraulics in an oxidation ditch, especially how the flow pattern, power output and velocity distribution are affected by the position and rotating speed of the submerged impeller. Stamou (2008) and Littleton et al. (2007) provided a design with an assumption of an ideal flow pattern, ignoring the real hydrodynamic characteristics of the ditch. With the development of computational fluid dynamics (CFD), flow patterns in oxidation ditches have been simulated by applying a 2D standard $k - \varepsilon$ turbulence model (Stamou 1993). Luo et al. (2004) present a 3D $k - \varepsilon$ turbulence model to simulate the flow field in an integrative oxidation ditch aerated with one set of brush aerators. Yang et al. (2010) proposed a moving wall model to simulate many sets of surface aerators with the velocity and momentum following the continuity and momentum equations. However, they cannot be relied upon for a complete simulation of flow field analysis because they only focused on evaluating the CFD model instead of
researching how the calibrated CFD model was conducted to compare and analyze the flow fields affected by different position and rotating speed of the power producer. To alleviate the problem, an experimentally validated CFD model was applied in this paper to compare and analyze the flow fields under different operating conditions of a submerged propeller.

Furthermore, the submerged impeller in the oxidation ditch is a major energy consumer of a wastewater treatment plant. Often more than one set of submerged impellers are in operation simultaneously to achieve the required flow velocities and process efficiency. Optimization of submerged impellers can reduce energy consumption and operation costs, as well as guarantee a reliable and efficient treatment in such installations with a certain number (Fayolle et al. 2007). Xie et al. (2014) used a commercial CFD package to study the flow field and sludge distribution of a full-scale oxidation ditch. Gancarski (2007) presented a single-phase model to simulate the flow field in an oxidation ditch by analyzing the flow in an independent model and then copying the velocity profile for the whole ditch simulation. Nevertheless, the above-mentioned literature rarely investigated the energy consumption of submerged impellers in the oxidation ditch, especially the oxidation ditches with many sets of impellers operating simultaneously. Wu et al. (2012) showed a model to predict the energy consumption based on CFD, but the simulated results were not found to be comparable with the measured data. Thus, a model of a certain number of submerged impellers will be applied to optimize energy consumption with the validation of experiments in this study.

The objective of this paper was to indicate an experimentally validated CFD model that can be used to optimize flow pattern and energy consumption by improving operating conditions of the full-scale oxidation ditch. A numerical model capable of predicting flow pattern and energy consumption was proposed for the oxidation ditch of the Yingtang wastewater treatment plant in Ma’anshan City in Anhui Province of China. Two cases were analyzed and compared by considering different installation position, rotating speed and number of submerged propellers, respectively. Both economic aspects and velocity distribution requirement were taken into account in optimization design. Based on the results of this study, several suggestions were put forward to optimize the performance of the oxidation ditch.

MATERIALS AND METHODS

Experimental set-up

The diagram of the ditch is shown in Figure 1 while Figure 2 presents the photos of field experiments. The channel is 12 m wide, with a water depth of 5 m and a central wall width of 0.25 m. The radius of the big semicircle is 6 m with the small semicircle of 3 m and the straight part of the channel is 120 m in length.

Measurements of the velocity were carried out at several sampling locations and different water depths with a cross-section size of 5 m × 9 m: the top layer 0.5 m below water surface, the middle layer 2.5 m below water surface, and the bottom layer 0.5 m from the ditch bed. Particularly, the flow along the semicircle portions is much more complicated than the straight portions because of the guide walls which make the quick change of circulating direction of the flow. Therefore, it is critical to subdivide the sampling locations in the semicircle portions so that the flow around can be analysed more accurately. The location details and numbering of sampling points are indicated in Figure 3(a) and 3(b). Horizontal flow velocities in the oxidation ditch were measured by an intelligent propeller current meter, with a range from 0.01 m/s to 2.0 m/s, which is manufactured by Nanjing R&D Technology Group Company of Nanjing Hydraulic Research Institute. The propeller-type current meter consists of a propeller, an optical fiber and a counting device. There are four blades including two reflective panels in each propeller, which gives two pulse signals after rotating for a circle. The velocity can be calculated by pulse numbers, recording time and calibration coefficient.

Figure 1 | Diagram of the oxidation ditch (unit: m). Numbers 1–12 refer to the respective cross-sections.
Two operating cases of the oxidation ditch were compared and analyzed: one was the case with only a submerged impeller but placed at different positions and worked with different rotating speed; the other was the optimized condition with three submerged impellers operating simultaneously.

**Mathematical modeling**

ANSYS fluid dynamics solutions are able to offer reliable results due to their solver robustness and advanced modeling capabilities. ANSYS renowned CFD analysis tools include the widely used and well-validated ANSYS CFX which can capture virtually any type of phenomenon related to fluid flow ([Ansly 2012](#)). Commercial code ANSYS CFX14.5 has been implemented for flow field and energy consumption computations in the CFD model. The CFD model typically consists of a description of flow geometry, a set of differential equations describing the physics of the flow, boundary and initial conditions, and mesh points at which these equations are solved ([Warsi 2005](#)). The software is based on the fundamental equations of continuity, momentum, and energy ([Littleton et al. 2007](#); [McClure et al. 2014](#)).

**Flow field modeling**

For the fluid flow analysis of the entire ditch, the continuity equation and Reynolds-averaged Navier–Stokes equation have been used in the following form ([Ruprecht 2002](#); [Qian et al. 2007](#); [Saeed & Galybin 2009](#)):

\[
\frac{\partial u_i}{\partial x_i} = 0 \tag{1}
\]

\[
\frac{\partial (U_i U_j)}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \nu \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \right] - \frac{\partial (\overline{u_i u_j})}{\partial x_j} \tag{2}
\]

where \( U, \rho, \nu \) and \( p \) are velocity, density, kinematic viscosity and pressure respectively. The renormalization group (RNG) \( k – \epsilon \) turbulence model is used for numerical predictions ([Sierra et al. 1999](#)). The simulation was conducted with the type of steady state.
The model of RNG $k-\varepsilon$ turbulence is capable of obtaining a more accurate description of the turbulent transfer relationship with the vortex scale changes or Reynolds number. Consequently, the model is better able to deal with the low Reynolds number zone or near the wall region (Ranade et al. 2001).

Computational domain

The computational domain was divided into two parts in order to rotate the turbine: an inner-rotating part and an outer-stationary which were defined as interface in CFX-Pre. The radius of each impeller is 0.9 m. The main function of the impeller is to pass energy to the fluid to obtain an axial flow velocity. A sliding-mesh approach was developed during simulation, with the radius of 1.0 m. The computed subject was just the same as the actual one with a working volume of 7,765 m$^3$ as shown in Figure 1.

Boundary conditions and wall functions

Stamou (1995) found that the inlet and outlet had little effect on the flow field in an oxidation ditch. As a consequence, the effects of inlet and outlet were not considered in the computation. The slip wall boundary condition was placed at the water surface. The no-slip boundary conditions were assigned for all the other walls, including the blade surfaces, the bottom surface, and the side and central walls of the ditch. And then the rotational speed, rotation-axis origin and direction of the moving wall were set, respectively. Residual error of CFX here was set as 10$^{-5}$. The reference pressure of the flow was 1 atm and there were no buoyant forces considered.

It is difficult to obtain a near wall cell that is well placed within the laminar sublayer (i.e. at $Y+ \approx 1$) where the boundary layer could be correctly resolved (McNaughton et al. 2012). $Y+$ is the dimensionless wall distance. The problem of inconsistencies in the wall function is able to be overcome with the use of the scalable wall function formulation developed by ANSYS CFX (Ansys 2012). The advantage of this formulation is that the near wall distance is not allowed to fall below a set limit (in CFX this is set at 11.06) so that all mesh points are outside the viscous sublayer and all fine mesh inconsistencies are avoided.

Grid independence

For the studied ditch, 3D grid meshes were created by using the ICEM CFD 14.5. Unstructured meshes were adopted due to irregular shape of the blade and many boundaries. Refinement of the mesh was carried out in order to produce a more accurate result by reducing residual errors. The number of cells was increased close to the walls of the blades where the influence of boundary layers would have greater effects on the flow. Convergence of the solution is sought for four levels of meshes including coarse, medium, fine and refined ones for the computational domain described in Figure 4. The fine mesh changed these values by less than one percent and so this level of refinement is not deemed necessary here and the cell number of 2.32 million was used taking into account accuracy and computational time. The calculation time was approximately 24 hours for a steady-state calculation on an Intel Core computer, with a 2.30 GHz i5 bi-processor and 6 GB RAM.

RESULTS AND DISCUSSION

There are two working conditions considered. In the first case, only submerged propeller A was in operation, whilst in the second one, the three propellers A, B, C worked simultaneously. All propellers were installed at mid water depth.

Case one

Performance and flow conditions due to the propeller position

1. Flow field. The submerged propeller A, as shown in Figure 5, was initially studied at three relative longitudinal
positions (5, 10 and 15 m) from the first curved wall. All profiles of velocity at the middle layer were predicted by taking into account the presence of the propeller. From the simulations, it was found that the assigned position of propeller A has a significant impact on the flow pattern, especially in the curved walls of the oxidation ditch. It was also seen that a heterogeneous distribution of flow field can be generated within the ditch.

From the three simulations, it was observed that a sludge deposit may occur in the guide walls (cross-sections 5 and 11) due to the small velocities. However, the highest discrepancy of propeller performance was observed at the guide wall described in cross-section 11. The flow with the propeller 5 m from the first curved wall demonstrated a better performance as the velocity near the external wall was obviously higher than the two other conditions and the flow overall was more homogeneous.

(2) Comparison of CFD results and experiments. Flow velocities were measured at the curved bend of the ditch (cross-section 11) using an intelligent propeller current meter. Velocity measurements were made with propeller A operation at the longitudinal locations indicated in Figure 3(b). The flow velocities under the three working conditions for the top, middle and bottom layer were plotted and compared against each other. A comparison was then made with the CFD predictions at given locations.

As shown in Figure 6(a), under the operating condition where propeller A was installed 5 m from the first curved wall, in the external area of section 11, for the bottom layer, the flow velocity rose from 0.145 to 0.163 m/s before it showed a considerable drop to 0.138 m/s as the absolute value of Y-axis decreased. For the middle and top layers, the flow velocities remained almost constant before they dropped to 0.136 m/s. In the internal area of section 11, the flow velocity grew from 0.069 to 0.120 m/s (bottom layer), 0.079 to 0.114 m/s (middle layer) and 0.074 to 0.113 m/s (top layer), respectively.

When propeller A was installed 10 m from the first curved wall, in the external area of section 11, for the bottom layer, the flow velocity increased from 0.130 to 0.139 m/s before it decreased to 0.120 m/s as the absolute value of Y-axis declined. For the middle and top layers, the flow velocities remained almost constant before they dropped to 0.120 m/s. In the internal area of section 11, the flow velocity grew from 0.065 to 0.110 m/s (bottom layer), 0.070 to 0.100 m/s (middle layer) and 0.064 to 0.097 m/s (top layer), respectively. When propeller A was installed 15 m from the first curved wall, the flow velocity distribution showed nearly the same tendency as the condition with the propeller installed 10 m from the first curved wall.

As we can see from Figure 6, the velocities in the external area of the curved wall were higher than those found in the internal area. This phenomenon was readily understandable because the fluid was dragged towards the external wall due to inertia and centrifugal force.

Most of the computational results match well with the measured data and the predicted flow field values were within $-1.28 \pm 7.14\%$ difference from the actual measured values, indicating that the flow pattern in the oxidation ditch with submerged impellers operating was able to be captured by the CFD model. The difference was probably due to the high suspended solids concentrations in the oxidation ditch because of the limitation of the single-phase approach.
Based on the simulation results and field measurements, the flow with the propeller 5 m from the first curved wall demonstrated a better performance, as the flow velocity near the external wall was obviously higher and more homogeneous. Therefore, this position is preferred since the faster flow at the top layer could spread into the middle and the bottom layer, which may lead to a more excellent performance in the curve channel (area of cross-section 11). The optimized location of the propeller could reduce energy loss caused by fierce collisions at the curved walls. Furthermore, the problem of sludge deposit and the low velocity in the bend that is caused by the attenuation of the drive power could be solved.

**Operation under different rotating speeds**

According to the empirical findings (Rongsen 2006; Wu et al. 2012), a propeller with the diameter of 1.8 m is capable of pushing flow for 80 m. Therefore, this finding applied to the experiments converts cross-section 9 into a suitable position for taking the measurements, since it is located 85 m away from propeller A and the flow in this area is homogeneous and stable.

The operating rotational speed of this kind of propeller ranged from 55.03 to 70.03 rev/min. The results of flow velocity distribution at the bottom layer with different rotating speed (5.76, 6.07, 6.50, 6.91 and 7.33 rad/s, respectively) are compared in Figure 7. The rotational speed against the required torque and power are as shown in Table 1.

As indicated in Figure 7, the velocity in the bottom layer met the requirement of 1.5 m/s when the rotating speed was greater than 6.50 rad/s. Meanwhile, as we can see from Table 1, the...
power consumption increased with the growth of rotating speed. Consequently, when the flow requirement and power consumption were considered together, the propeller with the rotating speed of 6.50 rad/s was obtained as the optimal option.

**Case two**

Based on the design procedure made by the company of LJM in Denmark, the minimum power consumption of the oxidation ditch to achieve the expected velocity could be calculated by the following formula, after the fluid medium and size of the ditch are determined (Rongsen 2006; Wu 2013).

\[
H_f = \left( \sum \varepsilon + \lambda \frac{UL}{F} \right) \frac{C^2}{2} \\
= \left[ 4 \times 1 + 0.5 \times \left( \frac{240 + 3.14 \times 6}{30} \times 16 \right) \times \frac{0.15^2}{2} \right] = 0.51 \text{ J/kg} \tag{3}
\]

\[
W_r = H_f \cdot Q \cdot \rho = 0.51 \times 0.15 \times 30 \times 1,000 = 2.30 \text{ kW} \tag{4}
\]

\[
W_R = W_r / \eta = 2.3 / 27\% = 8.52 \text{ kW} \tag{5}
\]

where \(H_f\), \(W_r\) and \(W_R\) represent hydraulic loss, calculated energy loss and energy consumption of the oxidation ditch, respectively; \(\varepsilon\), \(\lambda\), \(U\), \(L\), \(F\), \(C\), \(Q\), \(\rho\) and \(\eta\) represent local resistance factor, frictional resistant coefficient, wetted perimeter, midline length of ditch, discharge area, flow velocity, discharge in a time unit, density and efficiency, respectively.

On the basis of the calculation in case one, the flow pattern in the whole ditch with only a propeller in operation was not satisfied. As illustrated in Figure 5, the flow velocity distribution with propeller A in the majority of the area of the oxidation ditch was less than 0.15 m/s, which could lead to sludge settling and not meet the demanded criteria of the flow. Thus, propellers A, B and C were set up to operate simultaneously (case two), in order to achieve the required flow in the oxidation ditch.

As indicated in Figure 8, the whole oxidation ditch showed a satisfactory performance when three propellers operated at the same time. The flow distribution was homogeneous and the values in almost every section of the ditch were higher than 0.15 m/s.

Also, the distribution of the flow velocity against the lateral position was obtained at cross-sections describing the curve walls (sections 5 and 11) and right and left section of the mid-channel (sections 1 and 9 and 3 and 7, respectively).

By analyzing the flow, as it went from bottom left to right section of the channel (sections 7 to 9). The absolute velocity (Figure 9(b) right) changed from 0.25 to 0.45 m/s and decreased slightly due to friction drag. However at the right section of the channel, it reached a homogeneous state (Figure 9(a), right). For the curved walls (Figure 9(c)), the velocities in the external area were found higher than those in the inner area. The CFD results of the three layers at sections 1
Figure 9 | Flow velocity (m/s) versus different cross-sections. (a) Cross-sections 1 and 9; (b) cross-sections 3 and 7; (c) cross-sections 5 and 11.
and 3 demonstrated that the phenomenon of sludge settling would not occur due to the high predicted velocities.

The power and torque against rotational speed consumed for each propeller were calculated as shown in Table 2. For the second case, since the three sets of submerged propellers were operated at the same time, the total power reached a value of 8.89 kW. However, this power consumption was slightly higher than the minimum power of the oxidation ditch calculated before.

It is well-known that power consumption is an important evaluation index for pushing flow in the oxidation ditch. If the total power provided is less than the minimum power, the purpose of avoiding sludge deposit could not be realized. In contrast, if the total power provided is much greater than the minimum power, it may cause waste. Therefore, when the propelled water meets the requirement of the oxidation ditch, the less power that is consumed, the more fuel that is saved, which results in lower costs of operation. In case two, the submerged propellers could satisfy the need of the oxidation ditch using a power consumption that is almost above the minimum theoretical value. Therefore, this set-up was shown to be efficient.

**CONCLUSIONS**

An experimentally validated numerical tool capable of predicting flow pattern was developed for oxidation ditch systems by the 3D RNG $k – \varepsilon$ turbulence model. Two operating cases were compared and analyzed by considering two important factors: flow field and energy consumption.

The performance demonstration and comparison of different operating conditions were carried out in the full-scale oxidation ditch. The flow velocity at different measurement points was monitored and energy consumptions were calculated.

The following conclusions could be obtained from this study:

1. 3D RNG $k – \varepsilon$ turbulence model based on CFD could successfully characterize flow pattern and energy consumption in the full-scale oxidation ditch. The predicted flow field values were within $-1.28 \pm 7.14\%$ difference from the actual measured values. Thus, 3D CFD simulation could be a good supplement for optimization design of submerged propellers in oxidation ditches.

2. In case one, a submerged propeller should be relocated 5 m from the first curved wall with the rotating speed of 6.50 rad/s to achieve the minimum energy consumption while obtaining the desirable flow field distribution.

3. In case two, by determining three sets of submerged propellers after numerical simulation and actual measurement, not only the least power density but also the requirement of the flow pattern could be realized for the whole oxidation ditch.

However, the approach to treat suspended solid and water as a single phase might be limited. As a consequence, a two-phase CFD model should be modified to study the distribution of suspended solids under different operation parameters. The development of such a model is currently under investigation.

**ACKNOWLEDGEMENTS**

The research work was supported by the following funding: Research on Hydrodynamic Issues in Tidal Power (Chinese National Foundation of Natural Science-Key Projects, No. 51339005), the Fundamental Research Funds for the Central Universities (No. 2014B37614) and Science and Technology Project of Water Conservancy in Jiangsu Province (No. 2014026). The Chinese Scholarship Council supports the first author for studying in the University of Manchester (UK), where this paper was completed.

**REFERENCES**


---

<table>
<thead>
<tr>
<th>Number of submerged propellers</th>
<th>Rotational speed rad/s</th>
<th>Torque N·m</th>
<th>Power kW</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>6.50</td>
<td>470.89</td>
<td>3.06</td>
</tr>
<tr>
<td>B</td>
<td>6.50</td>
<td>445.90</td>
<td>2.90</td>
</tr>
<tr>
<td>C</td>
<td>6.50</td>
<td>450.43</td>
<td>2.93</td>
</tr>
</tbody>
</table>


Yang, Y., Yang, J., Zuo, J., Li, Y., He, S., Yang, X. & Zhang, K. 2011 Study on two operating conditions of a full-scale oxidation ditch for optimization of energy consumption and effluent quality by using CFD model. Water Research 45 (11), 3439–3452.