Flow field prediction in full-scale Carrousel oxidation ditch by using computational fluid dynamics
Yin Yang, Yingying Wu, Xiao Yang, Kai Zhang and Jiakuan Yang

ABSTRACT
In order to optimize the flow field in a full-scale Carrousel oxidation ditch with many sets of disc aerators operating simultaneously, an experimentally validated numerical tool, based on computational fluid dynamics (CFD), was proposed. A full-scale, closed-loop bioreactor (Carrousel oxidation ditch) in Ping Dingshan Sewage Treatment Plant in Ping Dingshan City, a medium-sized city in Henan Province of China, was evaluated using CFD. Moving wall model was created to simulate many sets of disc aerators which created fluid motion in the ditch. The simulated results were acceptable compared with the experimental data and the following results were obtained: (1) a new method called moving wall model could simulate the flow field in Carrousel oxidation ditch with many sets of disc aerators operating simultaneously. The whole number of cells of grids decreased significantly, thus the calculation amount decreased, and (2) CFD modeling generally characterized the flow pattern in the full-scale tank. 3D simulation could be a good supplement for improving the hydrodynamic performance in oxidation ditch designs.

Key words | Carrousel oxidation ditch, CFD, flow field, moving wall model

INTRODUCTION
The oxidation ditch is one of the widely used wastewater treatment reactors. There are several different kinds of oxidation ditches, such as integrative oxidation ditch, Orbal oxidation ditch, and Carrousel oxidation ditch. Correspondingly, there are different aeration equipments, for example, rotating brush, rotating disk, surface aerator. The common point among them is that they can bring oxygen into the oxidation ditch. The concentration of oxygen in the oxidation ditch changes with the distance to the aeration equipment. Thus aerobic, anaerobic and anoxic regions will exist in the oxidation ditch. With the presence of these regions, wastewater could be clarified during the circulation in the oxidation ditch. Wastewater will be purified only after circulating for a long time in the oxidation ditch, which makes oxidation ditches different from other wastewater treatment reactors. The biological processes in the ditch are reasonably well understood (Henze et al. 2000), and they have been captured and marketed commercially in packages such as GPS-x and Bio Win. Both of these packages are powerful tools to be used in the design and operation of treatment works to ensure the effective treatment. However, the hydraulics of the ditch are poorly understood due to the complex phenomena involved. Since the process efficiency depends on the flow field in the ditch, the design of the ditch cannot be fully successful unless realistic information on the hydrodynamics of the ditch is provided (Stamou 2007).

Nevertheless, with the improvement of computational power and the availability of CFD codes, the flow pattern in oxidation ditch has been studied. Researchers have done some investigations on hydrodynamic simulation in oxidation ditches. De Clercq et al. (1999), Stamou (1994, 1997), and Lesage et al. (2003) studied simple hydrodynamic features in the oxidation ditch by using a 1D model or
models with hydrodynamic effects. Stamou (1993) applied the 2D standard $k-\varepsilon$ turbulence model in an oxidation ditch simulation. In fact, it is more reasonable to use 3D $k-\varepsilon$ turbulence model to simulate the real flow field in an oxidation ditch. Luo Lin et al. (2005) used the 3D $k-\varepsilon$ turbulence model to simulate the flow field in a small-scale operational ditch (with a volume of 94 m$^3$, a length of 12 m, a width of 1.2 m, and depth of 3 m), and the simulated results were found to be acceptable by comparing with measured data. Deng Rongsen et al. (2005) used the 3D $k-\varepsilon$ turbulence mathematical model to simulate the flow field of rounded Orbal oxidation ditch. By comparison of the theoretical calculation data and measured results, it was indicated that the simulation results of mathematical model was reasonable. Unfortunately, the previously presented studies were limited to simulate only one set of brush or disc aerators. However, in oxidation ditches of real sewage treatment plant, more than six or seven sets of brush or disc aerators are in operation simultaneously to achieve the required process efficiency. Littleton et al. (2007) developed a CFD model by imparting the known momentum (calculated by tank fluid velocity and mass flow rate) to the fluid at the aeration disc region. The CFD model was validated with field data obtained from a test tank and a full-scale tank. The results indicated that CFD model could predict the mixing pattern in closed-loop bioreactors. But as the CFD model was developed first by imparting the known momentum to the fluid at the aeration disc region, it cannot be used for new tanks design where no field data are available. Gancarski (2007) used an one-phase and two-step model to simulate the flow field in oxidation ditch. The first step was to model the rotating brush and the second one was to model the whole ditch by using the results from the first step. The research provided a new way of simulating many sets of brush or disc aerators though it was essential to get more data in order to check the validity of the results.

The general objective of this paper was therefore to present an experimentally validated numerical tool that was able to predict flow field in Carrousel oxidation ditch with many sets of disc aerators operating simultaneously. A CFD model was developed for the Carrousel oxidation ditch of the Ping Dingshan Sewage Treatment Plant in Ping Dingshan City, a medium-sized city in Henan Province of China and the simulated results were compared with velocity data measured in the plant. The simulated results were found to be acceptable by comparing with measurement results. Moreover, based on the research, a possibly improving approach was proposed.

MATERIALS AND METHODS

Experimental setup

There are two Carrousel oxidation ditches in Ping Dingshan Sewage Treatment Plant. Both of them have the same configures and the wastewater flow rate of each ditch is 50,000 m$^3$/d. Therefore only one oxidation ditch was chosen for simulation. The full-scale, closed-loop bioreactor is a four-channel circular tank, with a total working volume of 26,000 m$^3$. The diagram of the ditch is shown in Figure 1. Each channel is 10 m wide, with a water depth of 4 m. The straight part of each channel is 133.750 m of length, and the radius of the big semi-circle part is 20.400 m and the radius of the small semi-circle part is 10.125 m. The central wall is 0.300 m in width with 10.150 m radius semi-circle ends. There are thirteen sets of disc aerators in the bioreactor. Each set consists of 45 discs that are 1.400 m in diameter.

In normal operating conditions part of each disc (about 0.500 m) is submerged under the wastewater surface. The operating condition with higher oxygen demand in the summer was tested with six aerator sets—two at 53 r/min, and the other four at 32 r/min.

Cross-sectional velocity in the full-scale facility were measured at two test locations in the straight part and the semi-circle part of the oxidation ditch (nine measurements per location), as shown in Figure 1. A one-dimensional portable propeller current meter (LS25-3, Chongqing hydrological instrument plant, Ministry of Water Resources of China, measurement range: 0.134–10.300 m/s) was used. The velocity probe was carefully oriented perpendicular to the flow during measurement, and the velocity values reported were the average of five to seven recordings. The velocity magnitude was probed at several sampling points, located at three different layers, the Top layer of 80 cm below water surface, the Middle layer of 160 cm below water surface, and the Bottom layer of 80 cm from the ditch bed. The location details are shown in Figure 2.
MATHEMATICAL MODELING

CFD modeling

Flow field computations were determined with the commercial CFD code FLUENT (version 6.3). This code uses the finite volume method for the discretization of the Navier-Stokes equations. The CFD models typically include a description of the flow geometry, a set of coupled differential equations describing the physics and chemistry of the flow, boundary and initial conditions, and a structured mesh of points at which these equations are solved (Anderson 1995; Warsi 1998). Segregated solver and the control-volume-based discretization scheme were used. The default parameter settings were used in FLUENT, as shown in Table 1.

Governing equations

The equations of motion in FLUENT are solved by a finite volume technique (Lesieur et al. 1996). The software is based on the fundamental governing equations of fluid dynamics, i.e. the continuity, momentum, and energy equations. These governing equations are programmed in FLUENT for conservation of mass and momentum, using the Navier-Stokes equations. Experience shows that the Navier-Stokes equations describe Newtonian flow accurately. Strictly speaking, “Navier-Stokes” refers only to the components of the momentum equation; however, it is a common artifice to include the continuity and energy equations in the set of equations referred by this term. There are several forms of the Navier-Stokes equations, depending on how they are derived (Littleton et al. 2007).

The Reynolds-averaged, Navier-Stokes equations, governing the three-dimensional, steady, incompressible flow in the Carrousel oxidation ditch as shown in Figure 1, were illustrated as follows:

Continuity equation:

$$\frac{\partial U_i}{\partial X_i} = 0,$$

Momentum equations:

$$\frac{\partial}{\partial X_i} (U_i U_j) = -\frac{1}{\rho} \frac{\partial P}{\partial X_i} + \frac{\partial}{\partial X_i} \left[ \nu \left( \frac{\partial U_i}{\partial X_i} + \frac{\partial U_j}{\partial X_j} \right) \right]$$

where the subscripts $i, j = 1, 2, 3$, $P$ is the pressure, $U_i$ is the velocity component in $i$ direction, $X_i$ is the coordinate component in $i$ direction, $\nu$ is the eddy viscosity, and $\rho$ is the density of water.

The $\nu$ in the above equations is determined with the $k-\varepsilon$ turbulence model (Rodi 1980):

$$\nu = C_{\mu} \frac{k^2}{\varepsilon}$$

where $\nu$ is related to the turbulent kinetic energy, $k$, and the dissipation rate, $\varepsilon$.

The distribution of $k$ and $\varepsilon$ are calculated from the following semi-empirical modeled transport equations:

$k$ equation:

$$U_i \frac{\partial k}{\partial X_i} = \frac{\partial}{\partial X_i} \left[ \nu \frac{\partial k}{\partial X_i} \right] + G - \varepsilon$$

$\varepsilon$ equation:

$$U_i \frac{\partial \varepsilon}{\partial X_i} = \frac{\partial}{\partial X_i} \left[ \nu \frac{\partial \varepsilon}{\partial X_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G - C_{2\varepsilon} \frac{\varepsilon^2}{k}$$

![Figure 1](https://iwaponline.com/wst/article-pdf/62/2/256/446289/256.pdf)

Figure 1 | Diagram of the computed oxidation ditch (unit: mm).

![Figure 2](https://iwaponline.com/wst/article-pdf/62/2/256/446289/256.pdf)

Figure 2 | Locations of velocity probe (unit: cm).
where $G$ is the production term of turbulent energy by the mean velocity gradients:

$$G = \gamma \left[ \frac{\partial U_i}{\partial X_j} \frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i} \frac{\partial U_j}{\partial X_i} \right]$$ (6)

The default values of the $k-E$ turbulence model constants are used (Launder & Spalding 1972), as shown in Table 2.

### Computational domain

The computed subject is an operational Carrousel oxidation ditch, with a working volume of 26,000 m$^3$, a width of 41.400 m and a depth of 4.800 m. The wastewater volumetric flow rate is 50,000 m$^3$/d, i.e. 0.579 m$^3$/s, and the normal cross-section area of single channel is 40 m$^2$. Therefore, the velocity of single channel contributed by inflow rate is 0.014 m/s, which is relatively small compared with the average velocity in the ditch and thus can be considered as steady flow. Regardless, Stamou (Stamou 1993) found that the inflow and outflow had little effect on the flow field in the ditch. Therefore, the inflow and outflow were not counted in the computation. The mechanical disc aerators were the only considered power source to drag the ditch flow. They maintained the movement of the mixture of liquid and suspended solid in the ditch, and provided intensive aeration to meet the oxygen demands of bio-chemical reactions in the wastewater treatment process. There are 6 sets of working disc aerators in the ditch, and 45 groups of rotation discs along the rotating shaft. The distance between discs in the same group is 0.200 m. The shaft axis is 0.200 m above the water surface, and the disc’s radius is 0.700 m. The computational domain was set up by using the pre-processor GAMBIT, illustrated in Figure 3.

### Aerators simulation

Stamou (1993) used the $x$-directional momentum component to model the rotation discs in his 2D simulation. In Luo Lin et al.’s (2005) model, moving meshes were created to directly reflect the drag in the fluid, and no other empirical approach was needed to model the power source. The moving meshes were built to encase the blade part submerged in the water. The sidewalls, the outer wall, and the surface of each zone formed a complete moving zone. Six moving zones were needed for 6 groups of blades. The sidewalls and the outer wall served as the interfaces between the moving and the fixed mesh, which were the main sources of the drag force. However, when this simulation method was applied to the oxidation ditch with many sets of operating brush or disc aerators, the number of interfaces and meshes was so large that the calculation amount increased dramatically.

In this study, there are 6 sets of working disc aerators in the ditch, and 45 groups of rotation discs along the rotating shaft for each set of disc aerator, as shown in Figure 1. If Luo Lin et al.’s (2005) model was applied in this studied oxidation ditch, namely one moving zone formed by two sidewalls, one outer wall and one surface

### Table 2 | $k$–$\varepsilon$ turbulence model constants

<table>
<thead>
<tr>
<th>$C_m$</th>
<th>$C_r$</th>
<th>$C_\omega$</th>
<th>$\alpha_k$</th>
<th>$\alpha_\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.00</td>
<td>1.50</td>
</tr>
</tbody>
</table>

---

The flow field prediction in full-scale Carrousel oxidation ditch by using CFD method is discussed. The computational domain was set up by using the pre-processor GAMBIT, illustrating the aerators simulation. Stamou (1993) used the $x$-directional momentum component to model the rotation discs in his 2D simulation. In Luo Lin et al.’s (2005) model, moving meshes were created to directly reflect the drag in the fluid, and no other empirical approach was needed to model the power source. The moving meshes were built to encase the blade part submerged in the water. The sidewalls, the outer wall, and the surface of each zone formed a complete moving zone. Six moving zones were needed for 6 groups of blades. The sidewalls and the outer wall served as the interfaces between the moving and the fixed mesh, which were the main sources of the drag force. However, when this simulation method was applied to the oxidation ditch with many sets of operating brush or disc aerators, the number of interfaces and meshes was so large that the calculation amount increased dramatically.

In this study, there are 6 sets of working disc aerators in the ditch, and 45 groups of rotation discs along the rotating shaft for each set of disc aerator, as shown in Figure 1. If Luo Lin et al.’s (2005) model was applied in this studied oxidation ditch, namely one moving zone formed by two sidewalls, one outer wall and one surface

### Table 2 | $k$–$\varepsilon$ turbulence model constants

<table>
<thead>
<tr>
<th>$C_m$</th>
<th>$C_r$</th>
<th>$C_\omega$</th>
<th>$\alpha_k$</th>
<th>$\alpha_\omega$</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.09</td>
<td>1.44</td>
<td>1.92</td>
<td>1.00</td>
<td>1.50</td>
</tr>
</tbody>
</table>
was created for one group of rotation disc, while the sidewalls and the outer wall served as the interfaces, then the whole number of interfaces would be 810. Moreover, the number of cells of grids would be too complicated for the calculation.

Under such circumstances, a new method called moving wall model was proposed. As shown in Figure 4, the part-cylindrical moving zone is 9.400 m in width and 0.800 m in radius. A complete moving zone was formed by the sidewalls, the outer wall, and the surface. Each rotation disc served as moving wall, which was the main source of the drag force. The fluid in each moving zone got the velocity and momentum from the moving walls firstly, and then the velocity and momentum was passed to the rest fluid in the oxidation ditch following the continuity and momentum equations. Each moving zone comprised 45 groups of rotating discs, and 6 moving zones were needed for 6 sets of disc aerators. The two sidewalls and the outer wall of each moving zone sever as the interfaces, so the whole number of interfaces was 18, which was much less than that of Luo Lin et al.’s (2005) model. The other seven non-working sets of disc aerators were considered as stationary walls, and the no-slip condition was set for them. In this case, the whole number of cells of grids decreased significantly and the calculation amount became acceptable.

**Boundary conditions**

The slip wall boundary condition was applied at the surface of the water, and the rigid-lid assumption was adopted. The working rotation disc was set as moving wall, where the rotational speed, rotation-axis origin and direction were set. Besides, the roughness constant of the moving wall was set as 1 and the roughness height was set as 0.020 m. For all the other walls, the no-slip boundary condition was supplied, including the bottom surface, the side and the central walls of the ditch. In order to simplify the computation, neither inflow nor outflow was considered in this case in terms of the Stamou’s work (Stamou 1993).

**Meshing**

For the studied tank, 3D grid meshes have been created (using the pre-processor GAMBIT). Because there are too many boundaries in the studied ditch and the shape of the ditch is irregular, unstructured meshes were used. The size of resulting grids information was summarized in Table 3. The grids have been refined near the moving wall for the rotation disc to simulate the flow characteristics precisely. The grid size was set to be sufficient to fulfill a grid-independent solution.

**RESULTS AND DISCUSSION**

The results for liquid velocity calculated by using CFD model were compared with the measurement results. Based on the numerical simulation and measurement results, detailed discussion followed and a possibly improved approach was proposed.

<table>
<thead>
<tr>
<th>Tank</th>
<th>Volume (m³)</th>
<th>Number of cells</th>
<th>Number of faces</th>
<th>Number of nodes</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>26,000</td>
<td>1,288,648</td>
<td>2,700,014</td>
<td>273,756</td>
</tr>
</tbody>
</table>
Liquid velocity profiles

The liquid velocity profiles have been measured in two test locations (as shown in Figure 1) using a mono-directional propeller current meter.

Figure 5 represents simulated streamlines in the tank with aeration. As can be seen, the streamlines in the aeration zone are denser than those in the anoxic zone. However, with aeration, the streamlines are deformed by gas produced by disc aerators. Zones of re-circulation due to the drag of the liquid are also observed.

Figure 6 represents simulated velocity contour for a distance from the bottom of the tank of 0.800 m (Bottom), 2.400 m (Middle) and 3.200 m (Top). The CFD model predicted that a heterogeneous flow pattern developed within the tank. In particular, a higher velocity was exhibited at the surface after passing the aerator, which then spiraled down to the bottom of the tank after going through the first turn and returned to the surface at the opposite side of the channel. This flow pattern was qualitatively captured by the CFD model.

From the simulation results in Figure 6, it could be seen that the fluid velocities at top of the tank were generally higher than those at bottom of the tank. However, it should be noted that the fluid velocity values in some places, especially in the anoxic zone, were so small that the suspended solid (SS) may be induced to settle down in the ditch. Submerged propeller is being proposed to prevent the settlement of SS in the ditch.
For two measurement locations, liquid velocity profiles matched quite well with CFD model predicted value.

For the Bottom of the tank at Location 1, the experimental liquid velocity decreased from 0.218 to 0 m/s with larger distance from the internal wall. The velocities in some points were so small that the mono-directional propeller current meter could hardly measure the values. The experimental heterogeneity of the local velocity was also well predicted by the model.

For the Middle of the tank at Location 1, the liquid velocity varied from 0.265 to 0.209 m/s with the distance from the internal wall. The liquid velocity profile was relatively homogenous on the measurement section and was well reproduced by simulation.

For the Top of the tank at Location 1, the liquid velocity varied from 0.220 to 0.242 m/s with the distance from the internal wall. The match between the measured data and the computational results was acceptable.

As seen from the data above, the velocity near the bottom of the tank was less than the velocity near the top of the tank. This phenomenon was easily understood for disc aerators were the main source to provide energy to the surface fluid and then the energy was passed down to the bottom fluid by turbulent diffusion.

For the Bottom of the tank at Location 2, the experimental liquid velocity increased from 0.099 to 0.248 m/s with the distance from the internal wall. The experimental local velocity was well predicted by the model.

![Figure 7](https://iwaponline.com/wst/article-pdf/62/2/256/446289/256.pdf)

Figure 7 | Liquid velocity (m/s) versus distance from internal wall (m) ■, experimental; — — —, CFD.)
For the Middle of the tank at Location 2, the liquid velocity varied from 0.120 to 0.235 m/s with the distance from the internal wall. The liquid velocity profile was well reproduced by simulation.

For the Top of the tank at Location 2, the liquid velocity varied from 0.154 to 0.231 m/s with the distance from the internal wall. The match between the measured data and the computational results was good enough to support the numerical modeling and the assumption made earlier.

In the figures, most of the computational results matched well with the measurement. For those discrepant points, they were located near eddy areas or the bottom of the tank where the velocity was very small. This means the velocity measurement in those points could be difficult and uncertain. A measurement error related to the position of the probe should be added to the sampling error which was indicated in Figure 7.

Mean liquid velocity

The good prediction of the local liquid velocities induced a reliable determination of the mean liquid circulation velocity, which was deduced by integration of the components of the velocity vector on the measurement surface. The experimental mean liquid velocity was calculated as the arithmetic average of the velocities measured on the experimental section. The results for measurement at Location 1 and 2 are summarized in Figure 8.

As shown in Figure 8, there was some difference between experimental and numerical data. This difference mainly resulted from the difficult estimation of the complicated flow field and fewer measurement points which was certainly insufficient to determine the mean velocity. A previous study (Fayolle 2006) showed that one measurement point per square meter of section was necessary to determine precisely the complicated flow field. The number of measurement points for measurement at Location 1 and 2 was less than Fayolle’s recommendation (Table 4). Therefore, more measurement points for validation will be improved in further study.

Besides, there could still be scope for improvement in flow field optimization and energy saving in oxidation ditch. Based on the fully understanding of 3D flow characteristics in a ditch, it is possible to optimize the flow field and save energy in oxidation ditch by the optimizing simulation and the application of control strategy for the different disc aerators start-stop in our further research.

CONCLUSIONS

A numerical tool to simulate flow characteristics in Carrousel oxidation ditch with many sets of disc aerators was presented. Moving wall model was used for simulating the disc aerators to create fluid motion in the tank. This consisted of building the part-cylindrical moving zones for the rotating discs of the aerators, creating moving walls to directly reflect the drag in the fluid and imparting the rotational speeds to the moving walls to create fluid motion in the tank. The velocity predictions in the tank showed a good fit with measured data, although they were slightly low near the top and the outer wall.

In general, both measured and predicted flow velocity profile data indicated that the velocity increased towards the
outer wall and decreased towards the bottom. The velocity distribution seemed to be related to tank size and configuration, disc submergence, and disc rotational speed. The results demonstrated that the CFD model qualitatively captured many aspects of the flow pattern in the full-scale tank, with some quantitative differences. In this work, CFD was used as a tool to predict the flow pattern in the tank. In general, CFD simulation is considered good within 10% error, while within 20% is considered acceptable (ANASYS 2002). The model can be refined with finer grids, and the way velocity imparted can be modified. However, accuracy is a trade-off with the complexity of the model and simulation time.

The following conclusions arise from this research:

(1) A new method called moving wall model could simulate the flow field in Carrousel oxidation ditch with many sets of disc aerators operating simultaneously. The whole number of cells of grids decreased significantly and the calculation amount was acceptable.

(2) CFD modeling generally characterized the flow pattern in the full-scale tank within a certain error. Velocities were not uniform across the tank cross-sectional area. Moreover, a complex flow pattern was indicated, consisting of a high velocity discharge from the aerator, flowing downward to the bottom of the curve in the tank, and then upward to the top of the tank on the side opposite of the aerator.

(3) 3D simulation should be a good supplement for improving the hydrodynamic performance in oxidation ditch designs. Based on the fully understanding of 3D flow characteristics in a ditch, it is possible to design the function zones in a ditch with new concepts and theories.

(4) The match between the measured data and the computational results is acceptable and good enough to support the numerical modeling and the assumption proposed earlier. The differences may arise from the difficult and uncertain measurement in the eddy areas or the bottom of the tank where the velocity was very small. Laser Doppler Velocimeter (LDV), Acoustic Doppler Velocimeter (ADV) etc. are being proposed, by which researchers can explore the flow phenomenon and validate the numerical simulation with much more accuracy. It is hopeful that numerical simulation can provide a powerful tool for oxidation ditch designs in further study.

ACKNOWLEDGEMENTS

The authors are grateful to the persons who were involved in the experimental measurements on the full-scale aeration tanks. The authors thank for the funds support from Ping Dingshan Sewage Treatment Plant.

REFERENCES

ANASYS 2002 Personal communication. CFX User Conference, Pittsburgh, Pennsylvania.


