Journal of Water Process Engineering 12 (2016) 120–126

Contents lists available at ScienceDirect

Journal of Water Process Engineering

journal homepage: www.elsevier.com/locate/jwpe

# Determination of diffuser bubble size in computational fluid dynamics models to predict oxygen transfer in spiral roll aeration tanks

Mitsuharu Terashima[a,*], Magnus So[a], Rajeev Goel[b], Hidenari Yasui[a]

[a] Faculty of Environmental Engineering, The University of Kitakyushu, 1-1 Hibikino, Wakamatsuku, Kitakyushu, Fukuoka 808-0135, Japan
[b] Hydromantis, Inc., 1 James Street South, Suite 1601, Hamilton, ON, L8P 4R5, Canada

ARTICLE INFO

*Article history:*
Received 8 December 2015
Received in revised form 30 June 2016
Accepted 11 July 2016

*Keywords:*
Wastewater treatment
Aerobic processes
Aeration
Oxygen transfer
Activated sludge
$K_La$

ABSTRACT

The aeration tank is one of the most important processes in the biological wastewater treatment and water reuse system to remove organic pollutants in wastewater. The prediction of oxygen transfer using the computational fluid dynamics (CFD) method is crucial for minimizing the energy consumption of the aeration tank, which is one of the highest energy consuming units in a wastewater treatment system. Bubble size ($d_B$), which is a critical parameter in the CFD simulations, was investigated for different types of diffusers. CFD calculations were conducted to simulate the hydraulics in different aeration tanks and the calculated values of the volumetric oxygen mass transfer coefficient ($K_La$) were compared with experimentally measured data. A total of 19 tanks containing clean water and 8 tanks containing wastewater with different diffuser types and aeration intensities were simulated. The $K_La$ values determined for these configurations were fitted using a calibrated $d_B$ value specified in the CFD simulations. The results of the calculations indicate that the coarse-bubble diffusers, fine-pore diffusers, and slitted membrane diffusers have bubble sizes of 7–8 mm, 5–6 mm, and approximately 3 mm, respectively. Additionally, CFD simulations were conducted to simulate the flow pattern and calculate the corresponding $K_La$ value when the diffuser configuration was changed.

## 1. Introduction

The aeration tank is one of the highest energy consuming units in a wastewater treatment plant. Computational fluid dynamics (CFD) models can allow for more efficient design of these units and help cut down energy consumption. The main role of the aeration tank in a wastewater treatment system is to mix and supply oxygen to the mixed liquor in the tank. The performance of the tank is significantly affected by its hydraulics. Mixing affects the transportation of oxygen and substrates that are utilized by the microorganisms. Previous researchers have utilized CFD calculations to investigate biochemical reactions in non-ideal hydraulic systems [1–3]. Littleton et al. [4] successfully modelled oxygen transfer in a full-scale, closed-loop bioreactor using DO source terms to describe the bubble-liquid mass transfer and sink terms to account for microbial activity. Oda et al. [5] calibrated a coefficient for oxygen transfer to match the experimentally observed overall volumetric oxygen transfer coefficient ($K_La$) for the whole tank at various air flow rates and successfully simulated the DO values in aeration tanks.

Oxygen transfer from the bubble to the liquid, represented by $K_La$, is a key hydraulic parameter. The oxygen transfer is considerably affected by flow patterns or hydraulics in the aeration tank [6]. The layout of the diffusers in the aeration basins affects the oxygen transfer performance [7]. Therefore, the modelling and simulation of oxygen transfer based on the flow patterns in the aeration tank have attracted much attention for a number of years.

Sekizawa et al. [8] introduced simple hydraulic models to predict the volume fraction of air in aeration tanks by assuming that the bubbles occupy the region just above the diffusers. By combining the predicted air volume fraction, classical penetration theory of Higbie (with a calibration coefficient, $\kappa$), and measured bubble size, the $K_La$ values were simulated well. The use of the CFD method for rigorous prediction of the air volume fraction resulted in the introduction of another calibration coefficient, $\kappa$ [9]. Cockx et al. [10] modelled $K_La$ using the air volume fraction calculated from CFD and the local mass transfer coefficient calculated using Higbie's equation, and validated the model by comparing the calculated numerical results to experimental data obtained in a laboratory-scale airlift reactor. Oxygen transfer in pilot and full-scale oxidation

---

* Corresponding author.
*E-mail addresses:* m-terashima@kitakyu-u.ac.jp (M. Terashima), m-so@kitakyu-u.ac.jp (M. So), Goel@hydromantis.com (R. Goel), hidenari-yasui@kitakyu-u.ac.jp (H. Yasui).

http://dx.doi.org/10.1016/j.jwpe.2016.07.001
2214-7144/

ditch-aeration tanks with ethylene-propylene diene rubber membrane (EPDM) diffusers was successfully simulated by a similar model, using measured bubble sizes [11]. The bubble size was found to be the dominant parameter determining oxygen transfer.

The conventional activated sludge process with a spiral roll diffuser layout is commonly used in large-scale municipal wastewater treatment plants and industrial wastewater treatment plants, although the oxidation ditch system is common in small-scale municipal wastewater treatment plants. In industrial wastewater treatment plants, an increase in the organic loading or nitrification requirements sometimes necessitates an increased oxygen supply. The CFD model should be capable of simulating the bubble distribution, if the additional aerators introduced to increase the oxygen supply cause complex flow in the aeration tanks. In addition, the combination of biochemical reactions and novel oxygen transfer models constructed using CFD are likely to simulate the effect of equipment modifications sufficiently.

Many types of diffusers already exist. Slitted flexible membrane-type diffusers have been widely used recently owing to their high oxygen transfer performance. However, diffusers of lower specifications, such as porous ceramics or perforated caps, have also retained popularity because of their robustness and endurance. In the case of these diffusers, adequate setting of the bubble size is required in order to use the CFD method to design the aeration process.

In the present study, CFD calculations have been performed to match the measured $K_L a$ values for various types of diffusers and to assign appropriate bubble sizes to the diffusers in the CFD models. Application of the CFD method to the installation of additional diffusers in an aeration tank has also been demonstrated.

## 2. Materials and methods

### 2.1. Hydrodynamics model

To describe the multiphase flow of liquids and air, the Eulerian-Eulerian approach was implemented. This is a well-known two-fluid model, and the governing set of equations consists of the continuity and momentum equations for each phase [12,13]. The CFD simulations were performed using the commercial software CFX 14.5 (ANSYS, Inc.). The interfacial transfer of momentum by the drag created by the air bubbles was described by the Grace model [14]. As the momentum transfer models exclude the drag force, the lift force model [15] and turbulence dispersion force model [16] were introduced, although the virtual mass force and wall lubrication [17] force were not included, as in a previous study [18].

Turbulence was modelled using the standard $k$-$\varepsilon$ model for the liquid phase. The enhanced turbulence diffusivity caused by bubbles was simulated by the model of Sato and Sekoguchi [19]. The densities of air and liquid were assumed to be constant at $1.185\,\mathrm{kg\,m^{-3}}$ and $1000\,\mathrm{kg\,m^{-3}}$, respectively. The viscosity of activated sludge was modelled from the sludge concentration using an equation by Bokil and Bewtra [20]. The calculated viscosity values ranged from 4.4 to 8.1 mPa in this study.

Two-dimensional cross-sectional geometry models were built to represent the spiral roll aeration tanks. The configuration of the aeration tanks is shown in Fig. 1. Hexahedral meshes were mainly generated using ANSYS meshing (ANSYS, Inc.) for the numerical calculations. To check the grid size sensitivity, a representative cell number, $Nc$, was defined as the height of the tank divided by the mean cell size. As the sensitivity test, the average volume fraction of tank 11 (tank height = 5000 mm) was calculated in the conditions of $Nc = 100$ (mean cell size = 50 mm) and $Nc = 125$ (mean cell size = 40 mm). The same average volume fraction value of 0.0029 with $Q_V/V = 1.03\,\mathrm{m^3\,m^{-3}\,h^{-1}}$ was achieved, which indicated that the calculated volume fraction is independent of cell size at $Nc = 100$, and the following simulations were conducted at this $Nc$. One of the meshes applied in the simulations is shown in Fig. 2.

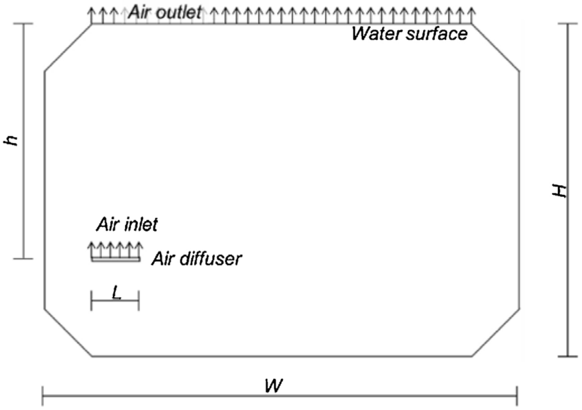

**Fig. 1.** Configuration of the aeration tank for the CFD calculations.

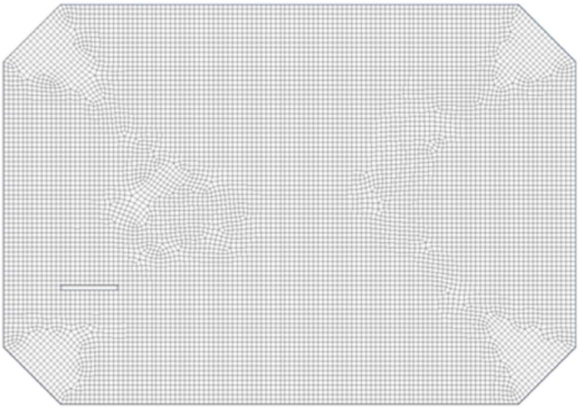

**Fig. 2.** Mesh applied in the simulations.

Generally, if there are differences between 2D and 3D simulations, then a 3D simulation is required [21]. To check the 2D model ability in this study, 3D and 2D simulations were compared. Considering the symmetry and periodicity of the layout, only part of the whole configuration was modelled in the 3D configuration. The configuration of tank 11 for the 3D simulation is shown in Fig. 3. The 2D and 3D simulations for $Q_V/V = 1.03\,\mathrm{m^3\,m^{-3}\,h^{-1}}$ give average air volume fractions of 0.0029 and 0.0030, respectively. As there is almost no difference between these simulations, the 2D model is a reasonable simplification for this configuration because this configuration has very high periodicity, which results in a flow pattern with very little flexibility. This phenomenon is very special for spiral flow aeration tanks in wastewater treatment plants, as the aerators are lined up in a row along a side wall, in contrast to an ordinal bubble column simulation.

A velocity inlet boundary condition was assigned for the air inlet on the top surface of the modelled diffuser, whereas the water surface was modelled under degassing conditions (i.e. only gases may go through the boundary and free slip is applied for the liquid phase).

Transient-type simulations were conducted in this study. After enough time had passed, the instantaneous results were used for the output. The time step was set at 0.1 s in all calculations. For the numerical method, the advection scheme was high resolution and the transient scheme was second-order backward Euler.