Chemical Engineering Science 109 (2014) 296–305

Contents lists available at ScienceDirect

Chemical Engineering Science

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

Simulation of flow field and sludge settling in a full-scale oxidation ditch by using a two-phase flow CFD model

Hao Xie [a], Jiakuan Yang [a,*], Yuchen Hu [a], Hao Zhang [a], Yin Yang [a,b], Kai Zhang [a], Xinfeng Zhu [a], Ye Li [c], Changzhu Yang [a]

[a] School of Environmental Science and Engineering, Huazhong University of Science and Technology, Wuhan, Hubei 430074, PR China
[b] School of Geography and the Environment, Oxford University, Oxford OX1 3QY, UK
[c] Universal Science & Technology (Shenzhen) Co., Ltd., Shenzhen, Guangdong 518057, PR China

HIGHLIGHTS

- A liquid–solid two-phase flow CFD model is set up in an OD.
- The average deviation is significantly lower than the result of single-phase model.
- An optimized operation scheme of OD is proposed based on the simulation.

ARTICLE INFO

Article history:
Received 23 July 2013
Received in revised form
23 January 2014
Accepted 2 February 2014
Available online 12 February 2014

Keywords:
Carrousel oxidation ditch
CFD
Hydrodynamic
Multiphase flow
Mathematical modelling
Dynamic simulation

ABSTRACT

A two-phase (liquid–solid) computational fluid dynamics (CFD) model for simulating the flow field and sludge settling in a full-scale Carrousel oxidation ditch (OD) has been proposed. The Takács double exponential sedimentation velocity function was applied to simulate the two-phase flow. The flow field simulation results are comparable with the data obtained from the field. Compared to the single-phase simulation, the relative error between the simulation results and field data in this two-phase model is reduced from 8% to 5%. Based on the simulation results of the flow field and sludge settling by using this two-phase CFD model, an optimized operation scheme of the OD was proposed. Compared with the existing one, the volume fraction of solid phase at the bottom of the OD in the optimized operation scheme decreases from 0.260 to 0.258, which seems to be insignificant, but the distribution of sludge becomes more uniform.

1. Introduction

With population growth in cities and rapidly developing economy, more and more wastewater is being generated in China. To deal with the situation, many wastewater treatment plants (WWTPs) have been built recently. In China, the most commonly used wastewater treatment process is oxidation ditch (OD): 32.1% of the WWTPs employ this process to treat wastewater (Yang et al., 2008). The popularity of the OD is mainly due to its reliability, simplicity of operation and good treatment performance. However, it also has some disadvantages. The operation consumes more energy and its suspended solid concentration is relatively high compared with other activated sludge processes (EPA, 2000; Grady et al., 2011), which poses a challenge for successful operation of the OD.

Flow and hydrodynamic characteristics play an important role in successful operation of an OD. It was difficult to study the rheology in a detailed way because it was time-consuming and expensive. However, due to the development of computational fluid dynamics (CFD), such a study has become easier and less expensive (Romaní Fernández and Nirschl, 2013; Stamou, 2008). CFD is a powerful tool to simulate the hydrodynamics and mass transfer and it has become increasingly popular in optimizing design and operation of WWTPs (Behin and Bahrami, 2012; Lesage et al., 2003; Mohajerani et al., 2012; Patel et al., 2011; Zhang et al., 2010). In CFD simulation of ODs, the common approach is to treat suspended solid and water as a single phase. Luo et al. (2005) used a 3-D k–ε turbulence model to simulate the flow field in a small-scale operational ditch, and the simulated results were found to be

* Corresponding author at: Huazhong University of Science and Technology, School of Environmental Science and Engineering, Luoyu Road 1037, Wuhan, Hubei 430074, PR China. Tel.: +86 27 87792207; fax: +86 27 87792101.
E-mail addresses: jkyang@mail.hust.edu.cn, yjiakuan@hotmail.com (J. Yang).

http://dx.doi.org/10.1016/j.ces.2014.02.002
0009-2509

comparable with the measured data. Gancarski (2007) used a single-phase model to simulate the flow field in an OD by analyzing the flow created by the surface aerator in an independent model and then copying the velocity profile for the whole ditch simulation. The settling of sludge by a particle tracking module was then investigated. The study was focused on the residence time distribution and the sludge concentration of suspended solids was not a parameter. However, the single-phase approach is not adequate because of the high suspended solids concentrations in ODs. In a single-phase model, a velocity profile needs to be established to show the velocity can keep the solids suspended. Although a single-phase model takes less time and predicts the flow well, the criterion for the minimum velocity is still in debate (Cumby, 1990; Cumby and Slater, 1990; Ebadian, 1998; Gunjal et al., 2005; Aubin et al., 2004). Not only the aqueous phase affects the flow, all the other phases influence the flow in an OD as well. Therefore, it is essential to include gas and solid phase in simulation. In wastewater treatment many liquid–solid phase models were applied to study the sludge settling in secondary clarifiers, while only few studies were focused on sedimentation occurred in biological reactors. Brannock (2003) used user-defined scalars to study the reaction and sludge settling in an anoxic wastewater treatment vessel. However, the scalar, defined as the sludge concentration, did not have any effects on the liquid phase, and the single-phase model was still used to predict velocity in the channel. In our previous studies (Yang et al., 2010, 2011), the flow field of an OD with disc aerators and submerged impellers was simulated well by using Moving Wall Model and Fan Model. In this study, a secondary solid phase was added to the model considered in our previous research, and a two-phase CFD model was modified to study the distribution of suspended solids. Subsequently the optimization of sludge concentration was studied under different operation parameters by using the modified two-phase CFD model.

The objective of this research is to study the flow field and sludge distribution of a full-scale oxidation ditch by using a commercial CFD package (Ansys Fluent 14). The schematic of this study is shown in Fig. 1.

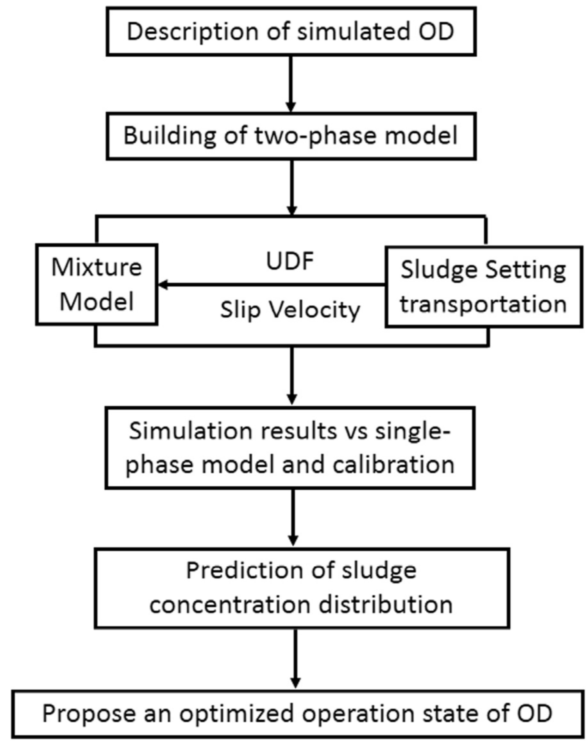

Fig. 1. The schematic of this study.

2. Methods

2.1. Full-scale OD description

The Carrousel 2000 OD simulated in this study is located in Ping Dingshan WWTP, Henan, China, with a treatment capacity of 50 000 m^3 per day. This full-scale bioreactor is a four-channel circular ditch, with a total working volume of 26 000 m^3. The mixed liquor suspended solid (MLSS) concentration in the OD is relatively high, ranging from 4000 mg/L to 6500 mg/L. There are 13 sets of surface aerators and 9 sets of submerged impellers in the bioreactor. The detailed information is shown in Fig. 2.

2.2. Mathematical equations

A solid–liquid phase three-dimensional (3D) CFD model was employed for prediction of flow speed and suspended solids distribution. The water-sludge system is considered as a two-phase flow using a full multi-fluid mixture model. The mass and momentum conservation equations were solved in the software.

Governing equations are the continuity equation and the momentum equation for the multi-phase flow. The continuity equation can be expressed as

$$\frac{\partial \rho_m}{\partial t} + \nabla \times (\rho_m u_m) = 0 \tag{1}$$

where u_m is the mass averaged velocity vector.

$$u_m = \sum_{k=1}^{2} a_k \rho_k u_k / \rho_m \tag{2}$$

where ρ_m is the mixture density.

$$\rho_m = \sum_{k=1}^{2} a_k \rho_k \tag{3}$$

where t, a_k, ρ_k, u_k are time, volume fraction, density and velocity vector of phase k (k=1, 2, for liquid phase and solid phase, respectively).

The momentum equation can be expressed as

$$\frac{\partial(\rho_m u_m)}{\partial t} + (u_m \times \nabla)\rho_m u_m = -\nabla p + \nabla \times \tau\rho_m + \rho_m g + \rho_m F + \nabla \times \left(\sum_{k=1}^{2} a_k \rho_k u_{d,k} \right) \tag{4}$$

where p, τ, g and F are pressure, stress tensor, gravity vector and body force, respectively, and $u_{d,k}$ is the drift velocity of phase k expressed as: $u_{d,k} = u_k - u_m$.

2.3. Suspended solid transport equation

The relative velocity (also referred to as the slip velocity) was used to describe the sludge settling. It is defined as the velocity of a sludge phase relative to the velocity of water phase in the vertical direction. Takács et al. (1991) double exponential sedimentation velocity function was applied.

$$v_s = v_2 - v_1 = \min |v_0 \times (e^{-k1 \times X} - e^{-k2 \times X}), Vs\ \max | \tag{5}$$

where v_s, v_0, v_1, v_2 Vs max, X are the settling velocity vector, the terminal settling velocity, the velocity of liquid phase, the velocity of solid phase, the maximum effective settling velocity, and MLSS concentration, respectively, and $k1$ and $k2$ are two constants.

In order to identify the parameters in settling velocity, the bench-scale experiment data collected from stirred cylinders with different suspended solids concentrations were used. The concentrations of

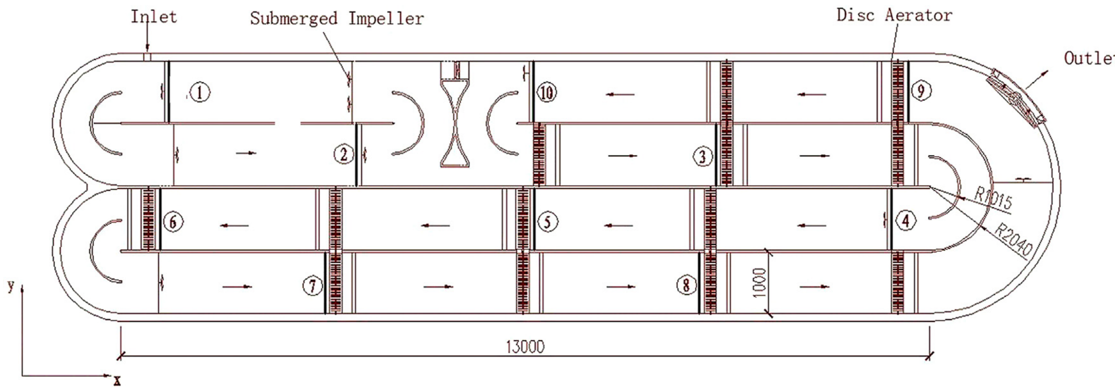

Fig. 2. The schematic diagram of the full-scale oxidation ditch. (TL: Test Location, unit: cm): ① TL1; ② TL2; ③ TL3; ④ TL4; ⑤ TL5; ⑥ TL6; ⑦ TL7; ⑧ TL8; ⑨ TL9 and ⑩ TL10.

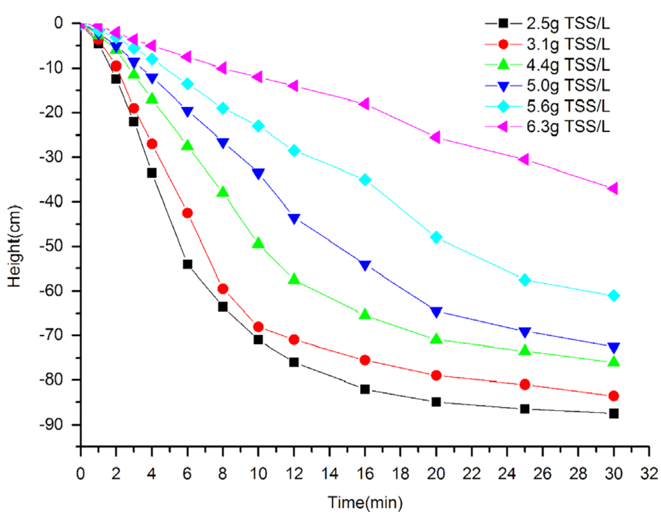

Fig. 3. The sludge settling curve for different concentrations.

Table 1
Settling velocities of MLSS vs. concentration of MLSS.

MLSS (g TSS/L)	2.5	3.1	4.4	5.0	5.6	6.3
Settling velocity ($\times 10^{-4}$ m/s)	18.92	13.54	9.17	5.83	3.96	1.88

the suspended solids were: 2.5, 3.1, 4.4, 5.0, 5.6 g/L and 6.3 g/L. The heights of the solid–liquid interfaces in the cylinders were measured as a function of time for 30 min. The results are shown in Fig. 3.

The settling velocities were determined from the slopes of the straight portions of the lines, and the results are shown in Table 1. The equation below represents the best fit between the setting velocities and MLSS concentrations.

$$v_s = 0.004 \times (e^{-0.46\times X} - e^{-1.86\times X}) \quad (6)$$

3. Numerical procedure

3.1. Geometry and grid generation

A model of the full-scale OD with multiple surface aerators and submerged impellers was built and meshed by Gambit, a general-purpose preprocessor for CFD analysis. The total computed volume of the model is 26 000 m^3. The detail of the geometry is shown in Fig. 2, with the computed domain shown in Fig. 4. Because of the complexity of the studied OD and its irregular shape, unstructured numerical grids were used. The grids near the surface aerators and submerged impellers were refined. After a series of grid-independent tests, the total number of elements turned out to be 1 526 308.

3.2. Boundary conditions

The boundary conditions play an important role in obtaining meaningful results in CFD simulations. There were thirteen groups of surface aerators and nine sets of impellers operating simultaneously. The operating forms of these devices are shown in Fig. 5. The devices were assumed to operate under steady state. Direct simulation of the devices needs a distinguished number of grids, which significantly increases the complexity of the CFD simulation. For simplification, a moving wall model was used to simulate the turning of the surface aerators, in which a complete moving zone was assumed to be formed by the sidewalls, the outer wall, and the surface. In addition, a fan model was adopted for simulation of the submerged impellers, in which each submerged impeller was assumed as an infinitely thin circle and the pressure difference across the impeller was calculated by the power input, area of the disc, density of the fluid and the average flow speed in the ditch. The detailed information can be found in our previous studies (Yang et al., 2010, 2011). A two-phase mixture model was used, and the slip velocity between sludge and water was employed as a user-defined function, in which Eq. (6) was adopted to simulate the settling of sludge. A user defined function (UDF) about the boundary conditions is present as supplementary information. The function of this UDF is to define the slip speed between solid phase and liquid phase. Velocity inlet and pressure outlet boundary conditions were applied and the concentration of suspended solid in the influent was used as the data measured at the OD inlet. The density of the dry sludge was assumed to be 1030 kg/m^3 and the volume fraction of the solid phase was obtained using this value.

4. Flow velocities and sludge concentration measurements

The actual liquid velocity profiles were measured at ten different locations (as shown in Fig. 2) using the mono-directional propeller flow meter (model: LS1206B; Nanjing Automation Institute of Water Resources and Hydrology, Nanjing, China) with a measurement range of 0.005–8.000 m/s. Suspended solid concentrations were measured by a portable suspended solids analyzer (model: PARTECH740; Partech Instruments, England) with a measurement range of 0–2000 mg/L and a temperature range of −10–60 °C. Measurements were taken at several locations and water depths. The details of the sampling locations are shown in Figs. 2 and 6.

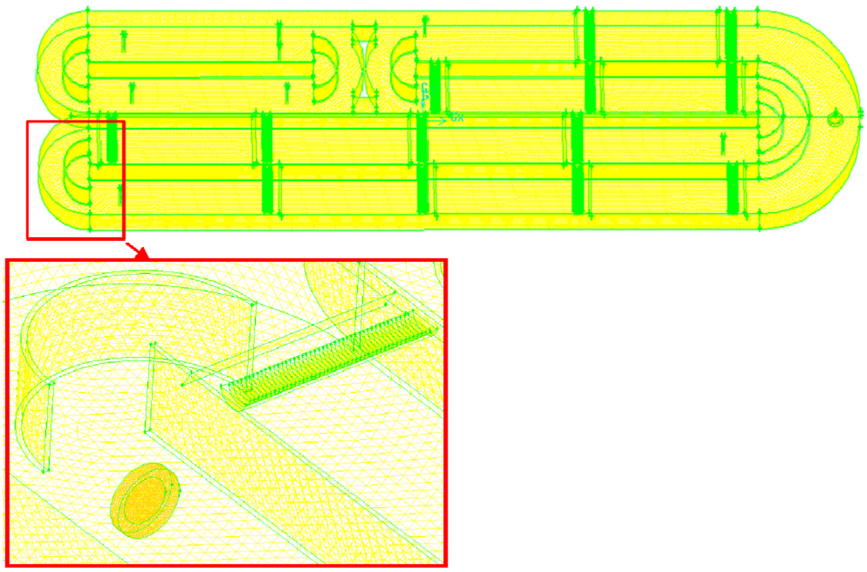

Fig. 4. The computed domain (modeled by Gambit with mesh.

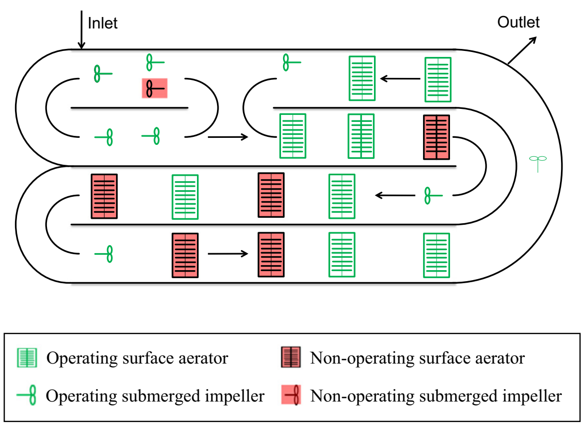

Fig. 5. Schematic diagram of OD operating condition.

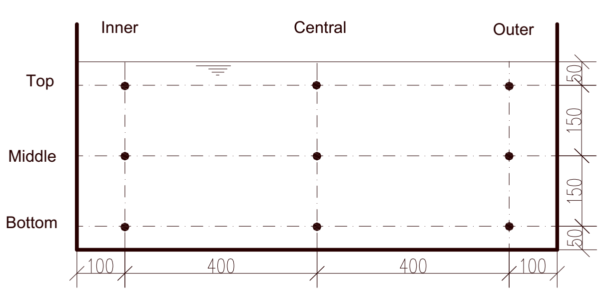

Fig. 6. Schematic diagram of sampling locations (unit: cm).

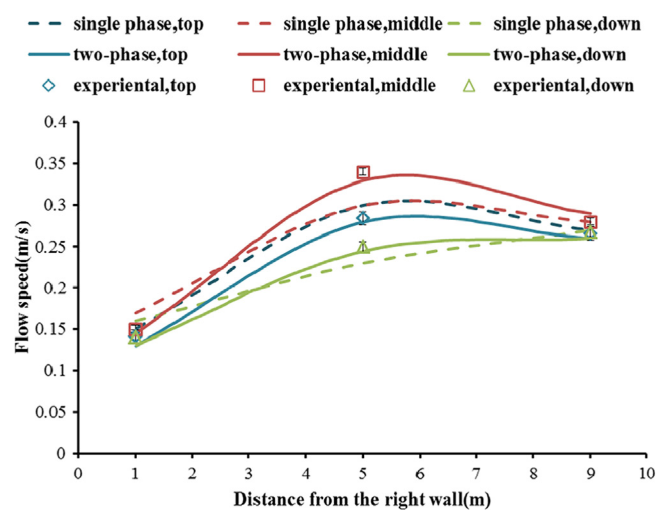

Fig. 7. Flow speed versus distance from the right wall at TL1.

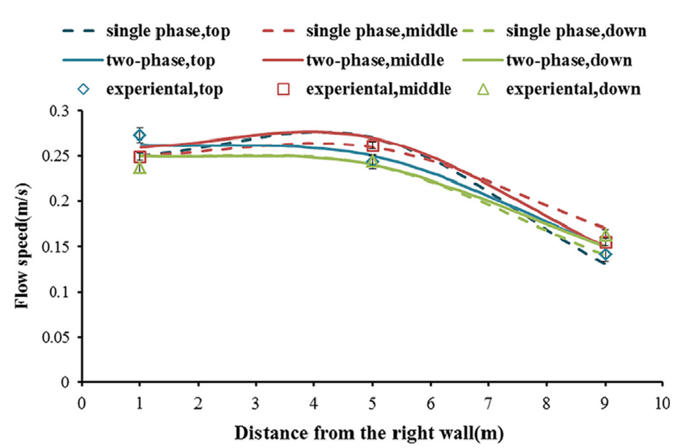

Fig. 8. Flow speed versus distance from the right wall at TL2.

5. Results and discussion

5.1. Velocity prediction by two-phase model comparison with single phase model

Comparison of the predicted average velocities by the two-phase model with those from the single-phase model and the measured field data are shown in Figs. 7–16. It can be seen from Figs. 7–10, velocities at the middle plane of the OD were higher than those in the top and bottom planes. It is because the submerged impeller was placed in the middle plane. At the drive of the impellers, the flow speed increased in the middle plane. Other figures show that the velocities at the bottom plane were the highest, and this might be caused by the guiding plate which guided the flow to the bottom of the OD, increasing the velocities at the bottom and reducing the possibility of sludge settling. From Figs. 7–16, it can be seen that the velocity profiles in the

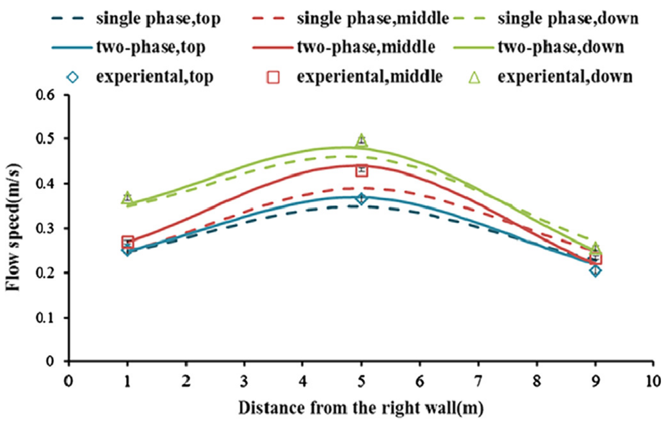

Fig. 9. Flow speed versus distance from the right wall at TL3.

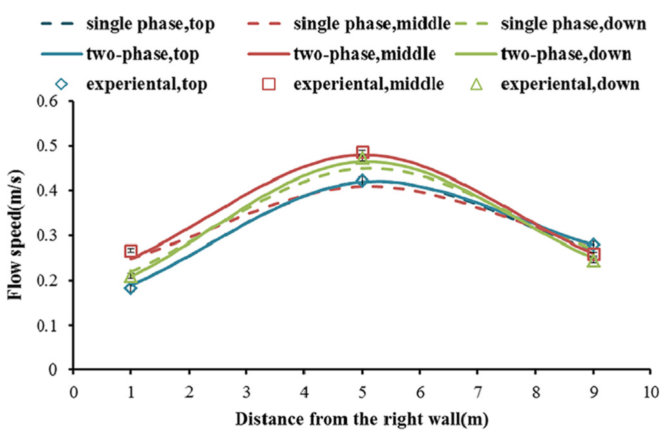

Fig. 10. Flow speed versus distance from the right wall at TL4.

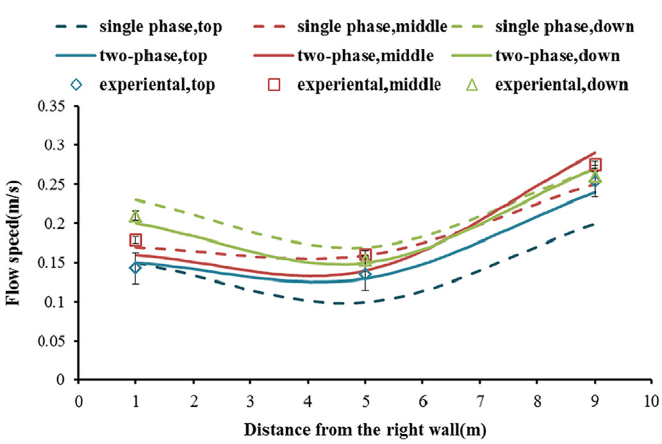

Fig. 11. Flow speed versus distance from the right wall at TL5.

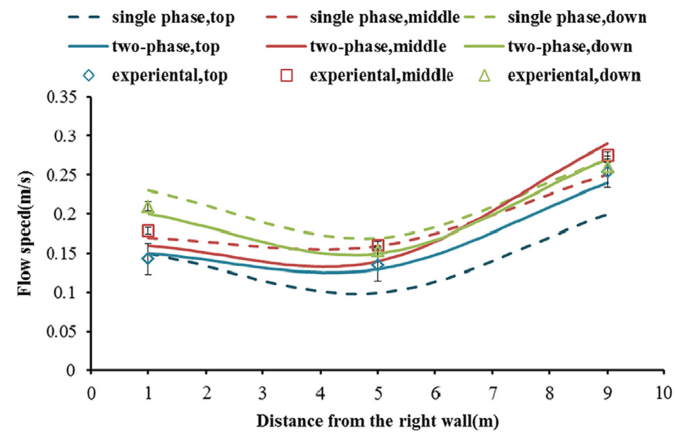

Fig. 12. Flow speed versus distance from the right wall at TL6.

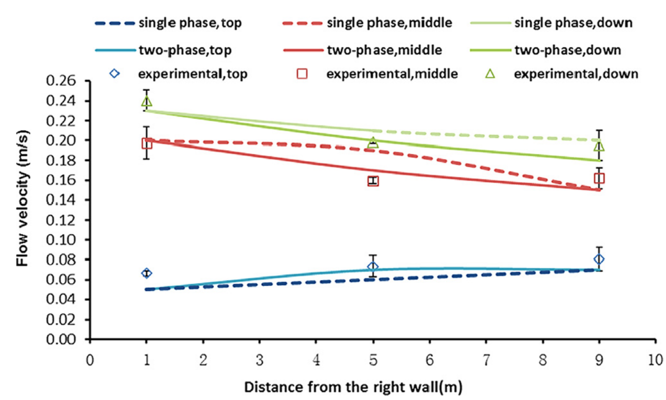

Fig. 13. Flow speed versus distance from the right wall at TL7.

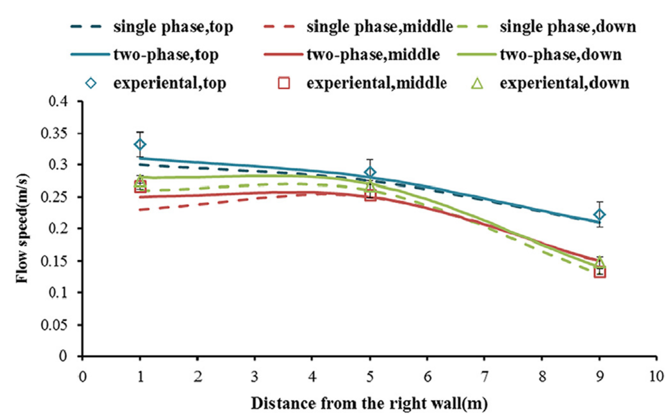

Fig. 14. Flow speed versus distance from the right wall at TL8.

single-phase and two-phase model are in a good agreement with measured data, but the two-phase model was better. The average deviation between flow velocities simulated by the two-phase model and the actual data is 5%, lower than 8% by using a single-phase model.

The contours of the velocity profiles at the top (0.5 m), middle (2.0 m) and bottom (3.5 m) of the OD predicted by the two-phase model are shown in Fig. 17. The average velocity at the top, middle and bottom are 0.176 m/s, 0.215 m/s, 0.226 m/s, respectively. These figures show that the highest velocity existed downstream of the aerators and the submerged impellers. The velocities at the bottom are lower than those at the middle and the top of the tank in the zone near the wastewater inlet, but the situation becomes opposite outside the wastewater inlet zone. This may be caused by the acceleration made by the submerged impellers and the diversion effects generated by the guiding plate located before the aerators. The higher velocities at the bottom of the OD can keep suspended solids. Velocities near the internal wall of the anoxic zone and at the big circular channel are relatively low at the top, middle and bottom planes as the change in shading in Fig. 17. Consequently, this may enhance sludge settling and reduce the effective treatment volume. To prevent the sludge settling from being enhanced, some optimization measures are needed. The optimization of sludge concentration distribution is discussed later.

Oxygen transfer terms could be added to the CFD model to determine if the DO concentration profiles could occur in the channel. A simple volumetric reaction can be considered.

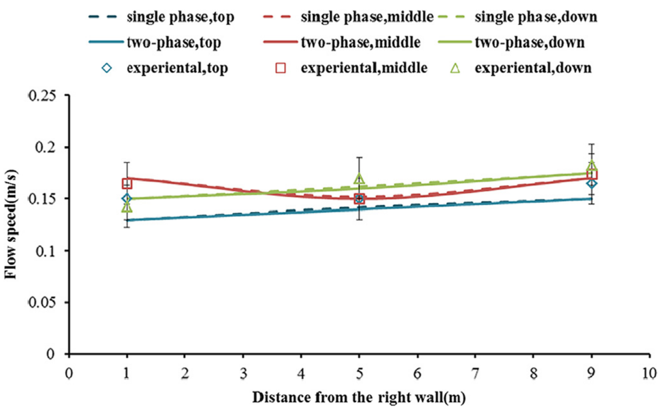

Fig. 15. Flow speed versus distance from the right wall at TL9.

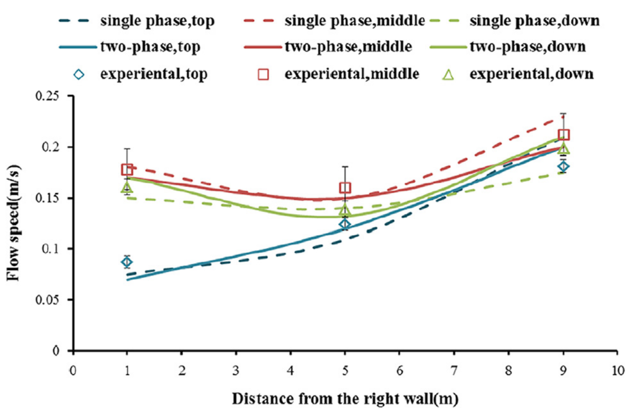

Fig. 16. Flow speed versus distance from the right wall at TL10.

Mass sources can be used to represent the surface aerators in operation. Surface aerators act as source regions and other regions act as sinks. The gas-phase dispersion transported by the turbulent fluid motion was taken into account in FLUENT by a user defined function (Littleton et al., 2007). The results of the model could provide an insight into the oxygen transfer in the ditch.

The mass transport equation is expressed as

$$\frac{\partial \alpha_k c_k}{\partial t} + \vec{\nabla} \times (\alpha_k c_k \vec{v_k}) = -\vec{\nabla} \times (\alpha_k (\vec{J_k} + \overline{c_k' v_k'})) + \overline{L_k} \tag{7}$$

where $\overline{L_k}$ represents the interfacial transfer of mass, J_k is the flux due to the molecular diffusion, α_k represents the fraction of phase k, c_k represents local instantaneous scalar the mass concentration in phase k, v_k represents local instantaneous phase velocity of phase k and $\overline{c_k' v_k'}$ represents the turbulent diffusion of the mass (Fayolle et al., 2007).

5.2. Distribution of suspended solids

The distribution of the suspended solids at the top, middle, and bottom of the OD are shown in Fig. 18. The average volume fraction of the top, middle, and bottom planes are 0.240, 0.249 and 0.260, respectively. It can be seen that the average volume fractions of the solid phase gradually increase from the top to the bottom of the OD. Comparing the velocity profile to the volume fraction of solid phase at bottom plane, it indicates that the concentration of the solid phase is inversely related to the velocity in the field, and the volume fraction of the solid phase increases when the velocity decreases. Most of the solids accumulate at locations where the velocities are relatively low. This indicates that the slow flow speed in the OD might cause the settling of suspended solid. It is important to note that the volume fraction of solid phase near the internal wall is higher than that near the external wall at the circular channel, which can be explained as follows: first it is because the velocity of the solids near internal wall is slower than that near the external wall at

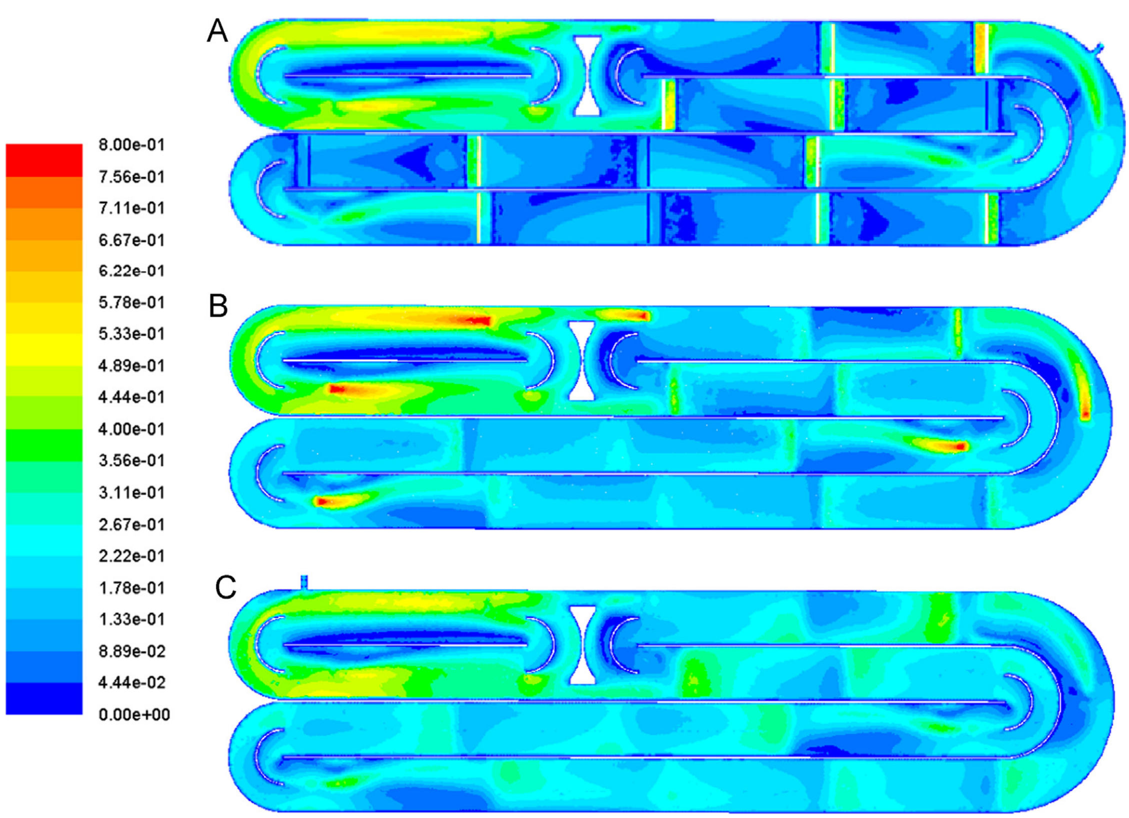

Fig. 17. Velocity contour profiles of horizontal section of the tank at different height (m/s): (A) top of the tank, (B) middle of the tank, and (C) bottom of the tank.

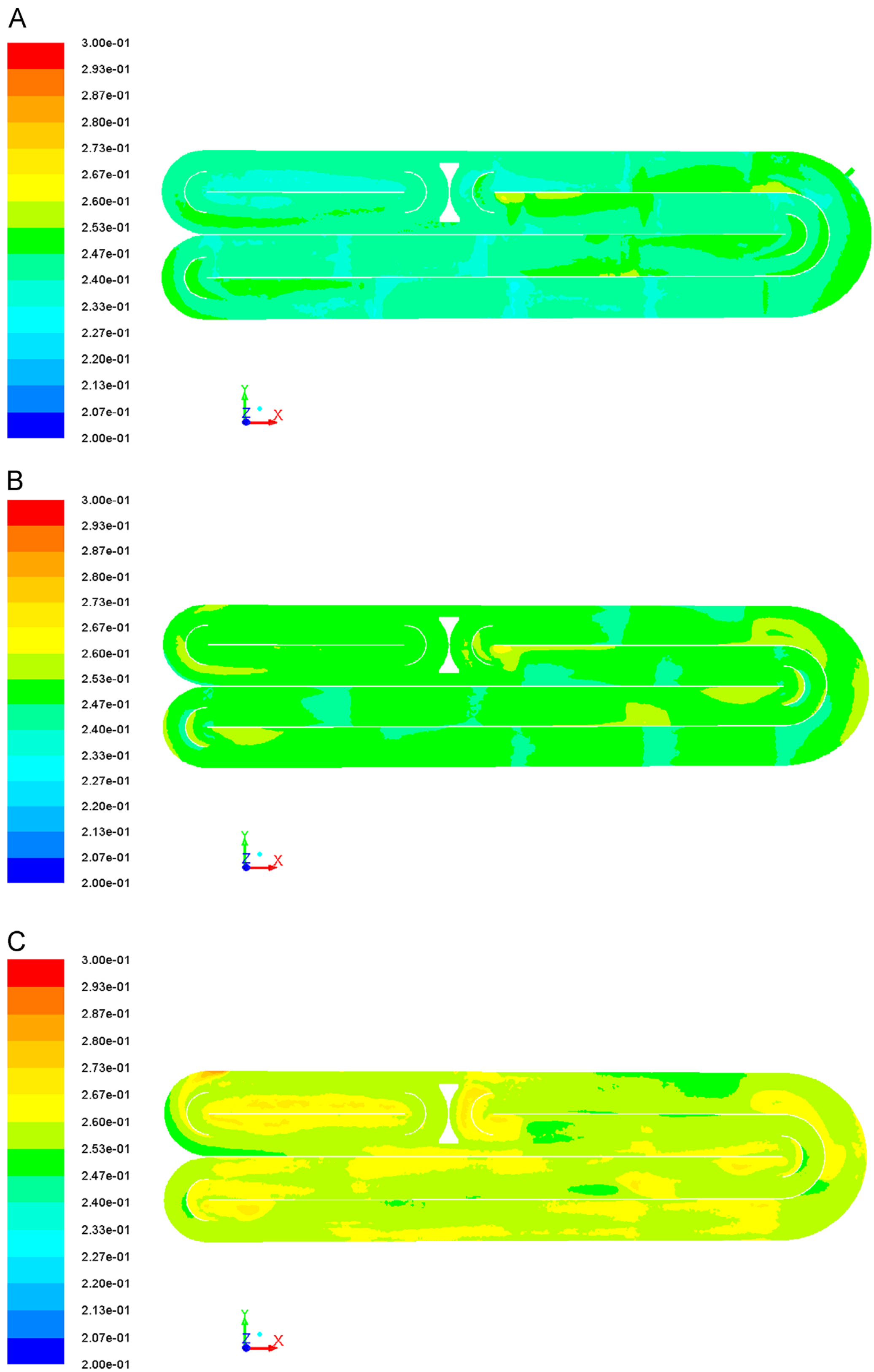

Fig. 18. Contours of volume fraction of solid phase at different height: (A) top of the tank, (B) middle of the tank, and (C) bottom of the tank.

the big circular channel; second it is probably because when the flow from the straight channel enters the circular channel, with the combined action of the centrifugal, gravity and counter-acting force generated by external wall, the suspended solids are pushed from the external wall toward the internal wall.

Based on the simulated results of the distribution of solid phase under the current operation shown in Fig. 5, an optimal scheme was proposed for the operation (Yang et al., 2011), as shown in Fig. 19. Compared with Fig. 5, the changes on the operation of the submerged impellers include: (1) turn-on of the submerged impeller near the internal wall in the anoxic zone, and (2) turn-off of the submerged impeller near the test location 1 (called as TL1). The changes are based on the fact that, the solids easily agglomerates near the internal wall, and the operation of the submerged impeller can accelerate the flow to reduce the potential of sludge settling. The changes made on operation of the aerators include (1) turn-off of the aerator at the entrance of the big circular channel; and (2) turn-on of the aerator in the fourth channel after a submerged impeller. The changes are based on the fact that the velocities in the fourth channel were relatively low and the flow speed at the circular channel entrance was requested not to be too high. These changes would not increase the energy consumption. After the changes were made, another model simulation for the new operation scheme was performed, by focusing on the bottom of the OD where it is most prone to sludge settling. The simulated results can be found in Fig. 20. The average volume fraction of solid phase at the bottom plane is 0.258, essentially the same as the current operation condition (0.260), but the distribution of solid phase becomes more uniform as shown in Fig. 20 when comparing to Fig. 18(C). The volume fraction near the internal wall decreases significantly; however, the solid at the exit of the circular channel is still higher than those at the other locations.

Three different points at the bottom plane were obtained to compare the effect of the sludge setting on original scheme with the optimal scheme. The locations of three representative points are present in Fig. 21. Point A, B and C were collinear, and their distances from the internal wall were 17.74 m, 13.31 m and 2.67 m, respectively. The volume fraction of solid phase is lower in optimal scheme compared with original scheme as shown in Fig. 22. Calculating the proportion of the whole OD area for which the volume fraction of solid phase is higher than 0.260, optimal scheme is 19.14% while the original scheme is 31.94%. The sludge settling problem is managed better in the optimal scheme compared with the original scheme.

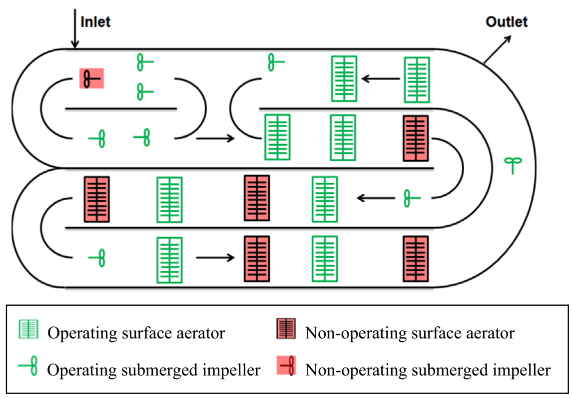

Fig. 19. Schematic diagram of optimized OD operating condition.

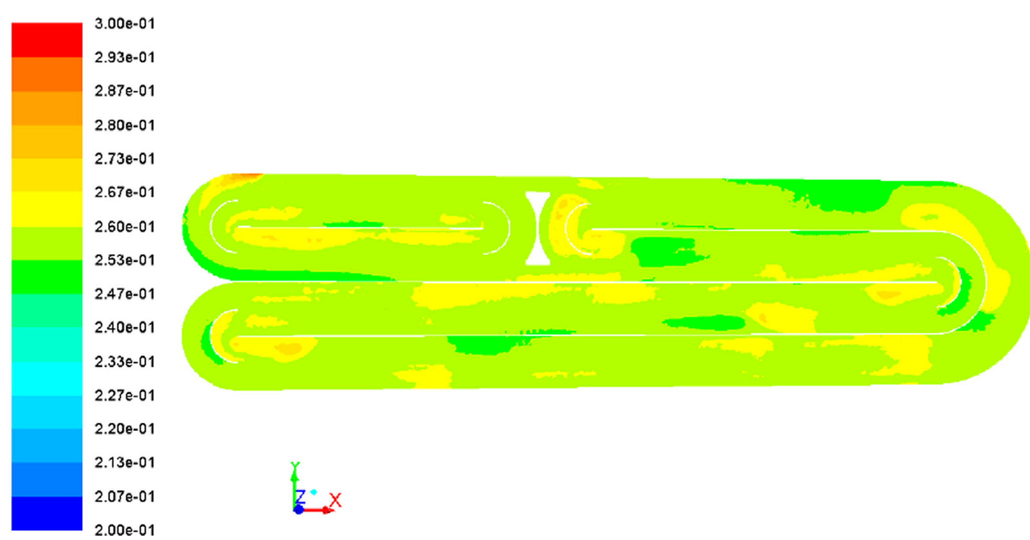

Fig. 20. Contours of volume fraction of solid phase at the bottom of OD.

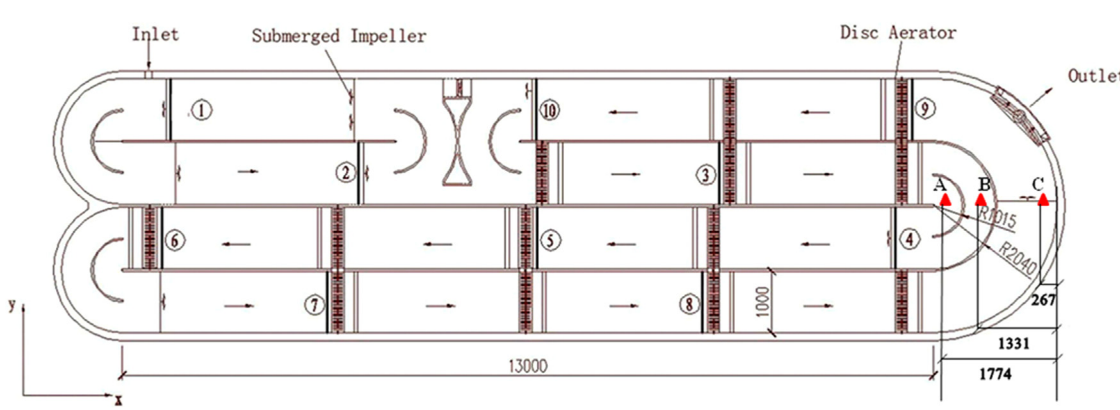

Fig. 21. Location of points A, B and C (Point A: at the distance of 17.74 m from the internal wall; Point B: at the distance of 13.31 m from the internal wall; Point C: at the distance of 2.67 m from the internal wall).

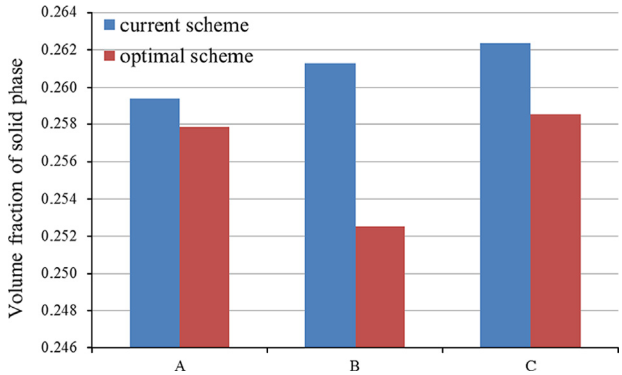

Fig. 22. Proportion of the volume fraction of solid phase in A, B and C points (Point A: at the distance of 17.74 m from the internal wall; Point B: at the distance of 13.31 m from the internal wall; Point C: at the distance of 2.67 m from the internal wall).

6. Conclusions

In this study, a liquid–solid two-phase flow CFD model was set up to study the velocity profile and suspended solids distribution in an OD. The average deviation between flow velocities simulated by the two-phase model and the actual data is 5%, lower than 8% by using a single-phase model. Modeling approaches of sludge settling in ODs were presented. Settling velocities derived from cylinders were used for modeling the settling behavior of the sludge. Based on the simulation results of suspended solid distribution, an optimized operation scheme was proposed. The distribution of sludge became more uniform and the concentration of suspended solids near the interior wall became smaller than that in the current operating mode with the application of optimized scheme. Delineating the solid concentration and velocity profiles by using two-phase flow CFD model can help optimization of design and operation of ODs.

Nomenclature

a_k	volume fraction, %
CFD	computational fluid dynamics
DO	dissolved oxygen, mg O_2/L
F	body force, N
g	production term of turbulent energy by the mean velocity gradients, kg/m s^3
MLSS	mixed liquor suspended solids, g TSS/L
OD	oxidation ditch
P	pressure, Pascal
R	radius of cylinder, m
t	time, s
T	temperature of the mixed liquor, °C
TN	total nitrogen, mg N/L
TSS	total suspended solids, g
u_m	mass averaged velocity vector, m/s
u_k	velocity component in k phase, m/s
$u_{d,k}$	the drift velocity of phase k expressed as: $u_{d,k} = u_k - u_m$
v_s	settling velocity vector, the velocity of liquid phase, m/s
v_0	the terminal settling velocity, m/s
v_1	the velocity of liquid phase, m/s
v_2	the velocity of solid phase, m/s
Vs max	the maximum effective settling, m/s
WWTP	wastewater treatment plant
X	MLSS concentration, mg/L
z	vertical distance from the water surface, m

Greek letters

ρ	density, kg/m^3
ρ_k	density of phase k, kg/m^3
ρ_m	mixture density, kg/m^3
τ	stress tensor, N/m^2

Acknowledgement

The authors are grateful to those who conducted the field measurements at the Ping Dingshan WWTP. The authors thank for the financial supports from New Century Excellent Talents Project of Ministry of Education (NCET-09-0392), Research Project of Chinese Ministry of Education (113046A) and National Natural Science Foundation of China (51078162). The authors would also like to thank the Analytical and Testing Center of Huazhong University of Science and Technology for providing facilities for experimental measurements.

Appendix A. Setting velocity user defined function edited

```
#include "udf.h"
#include "sg_mphase.h"
DEFINE_VECTOR_EXCHANGE_PROPERTY(custom_slip,c,
  mixture_thread,second_column_phase_index,
  first_column_phase_index,vector_result)
{
real c_sludge, v, change;
c_sludge=C_VOF(c,st) * 1050;
pgrad_x=C_DP(c,mixture_thread)[0];
pgrad_y=C_DP(c,mixture_thread)[1];
v=-0.004 *(exp(-0.46 * c_sludge)-exp(-1.86*c_sludge));
if (v>=0.0018902)
{change=0.0018902;}
else if (v<0.0018902)
{change=v; }
vector_result[2]=change;
}
```

References

Aubin, J., Fletcher, D.F., Xuereb, C., 2004. Modeling turbulent flow in stirred tanks with CFD: the influence of the modeling approach, turbulence model and numerical scheme. Exp. Therm. Fluid Sci. 28, 431–445.
Behin, J., Bahrami, S., 2012. Modeling an industrial dissolved air flotation tank used for separating oil from wastewater. Chem. Eng. Process. 59, 1–8.
Brannock, M.W.D., 2003. Computational Fluid Dynamics Tools for the Design of Mixed Anoxic Wastewater Treatment Vessels. The University of Queensland, Australia.
Cumby, T.R., 1990. Slurry mixing with impellers: Part 1, Theory and previous research. J. Agric. Eng. Res. 45, 157–173.
Cumby, T.R., Slater, N.K.H., 1990. A model of the vertical distribution of solids during slurry mixing. J. Agric. Eng. Res. 45, 187–207.
EPA, 2000. Wastewater Technology Fact Sheet: Oxidation Ditches, p. 〈http://water.epa.gov/scitech/wastetech/upload/2002_06_28_mtb_oxidation_ditch.pdf〉.
Ebadian, M., 1998. Improved Mixing and Sampling Systems for Verification Melted Feeds. Florida International University, United States.
Fayolle, Y., Cockx, A., Gillot, S., Roustan, M., Héduit, A., 2007. Oxygen transfer prediction in aeration tanks using CFD. Chem. Eng. Sci. 62, 7163–7171.
Gancarski, P., 2007. CFD Modelling of an Oxidation Ditch. Cranfield University, England.
Grady, C.P.L., Daigger, G.T., Love, N.G., Filipe, C.D.M., 2011. Biological Wastewater Treatment, third ed. The Chemical Rubber Company Press, Florida.
Gunjal, Prashant R., Ranade, Vivek V., Chaudhari, Raghunath V., 2005. Computational study of a single-phase flow in packed beds of spheres. Fluid Mech. Transp. Phenom. 51, 365–378.

Lesage, N., Spérandio, M., Lafforgue, C., Cockx, A., 2003. Calibration and application of a 1-D model for oxidation ditches. Chem. Eng. Res. Des. 81, 1259–1264.
Littleton, H.X., Daigger, G.T., Strom, P.F., 2007. Application of computantional fluid dynamics to closed-loop bioreators: I. Charateriztion and simulation of fluid-flow pattern and oxygen transfer. Water Environ. Res. 79, 600–612.
Luo, L., Li, W.M., Deng, Y.S., Wang, T., 2005. Numerical simulation of a combined oxidation ditch flow using 3D k-epsilon turbulence model. J. Environ. Sci. 17, 808–812.
Mohajerani, M., Mehrvar, M., Ein-Mozaffari, F., 2012. Photoreactor design and CFD modelling of a UV/H_2O_2 process for distillery wastewater treatment. Can. J. Chem. Eng. 90, 719–729.
Patel, T., Gill, L., Faram, M., 2011. Grit removal from wastewater using secondary currents in open-channel flow around bends. J. Environ. Eng. 137, 1026–1039.
Romaní Fernández, X., Nirschl, H., 2013. Simulation of particles and sediment behaviour in centrifugal field by coupling CFD and DEM. Chem. Eng. Sci. 94, 7–19.
Stamou, A.I., 2008. Improving the hydraulic efficiency of water process tanks using CFD models. Chem. Eng. Process. 47, 1179–1189.
Takács, I., Patry, G.G., Nolasco, D., 1991. A dynamic model of the clarification-thickening process. Water Res. 25, 1263–1271.
Yang, L.B., Zeng, S.Y., Ju, Y.P., He, M., Chen, J.N., 2008. Statistical analysis and quantitative recognition of energy consumption of municipal wastewater treatment plants in China. Water Wastewater Eng. (in Chinese) 34, 42–45.
Yang, Y., Wu, Y.Y., Yang, X., Zhang, K., Yang, J.K., 2010. Flow field prediction in full-scale Carrousel oxidation ditch by using computational fluid dynamics. Water Sci. Technol. 62, 256–265.
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 Res. 45, 3439–3452.
Zhang, D.J., Guo, L.S., Xu, D.Y., Chen, Y., 2010. Simulation of component distributions in a full-scale carrousel oxidation ditch: a model coupling sludge-wastewater two-phase turbulent hydrodynamics with bioreaction kinetics. Environ. Eng. Sci. 27, 159–168.