Journal of Environmental Sciences Vol.17, No.5, pp.808—812, 2005

ISSN 1001—0742
CN11—2629/X

Article ID: 1001-0742(2005)05-0808-05 **CLC number**: X131.2; X703 **Document code**: A

Numerical simulation of a combined oxidation ditch flow using 3D k-ε turbulence model

LUO Lin[1], LI Wei-min[2], DENG Yong-sen[2], WANG Tao[2]

(1. State Key Laboratory of Hydraulics, Sichuan University, Chengdu 610065, China. E-mail: luolin@tsinghua.org.cn; 2. College of Municipal Construction and Environmental Engineering, Chongqing University, Chongqing 400045, China)

Abstract: The standard three dimensional(3D) k-ε turbulence model was applied to simulate the flow field of a small scale combined oxidation ditch. The moving mesh approach was used to model the rotor of the ditch. Comparison of the computed and the measured data is acceptable. A vertical reverse flow zone in the ditch was found, and it played a very important role in the ditch flow behavior. The flow pattern in the ditch is discussed in detail, and approaches are suggested to improve the hydrodynamic performance in the ditch.

Keywords: oxidation ditch; 3D simulation; turbulence flow

Introduction

As being a high performance, space-saving wastewater treatment facility, the combined oxidation ditch has significant advantages for small residential communities. The whole wastewater treatment process, including aerobic bio-chemical degradation, solid-liquid separation, active sludge recirculation, etc., can be finished in one tank. The flow in the ditch is driven and aerated by the rotor, and no other external forces are needed. The ideal flow pattern in an oxidation ditch is plug flow and no recirculation, and velocity should be greater than 0.3 m/s to avoid solid material settling. Since the flow pattern in the ditch is very complicated, its hydrodynamic characteristics play a very important role in the design. Researchers have done some investigations on hydrodynamic simulation in oxidation ditches. De Clercq *et al*. (De Clercq, 1999), Stamou (Stamou, 1994; 1997), and Lesage *et al*. (Lesage, 2003) studied simple hydrodynamic features in the oxidation ditch by using a 1D model or models with hydrodynamic effects. Stamou(Stamou, 1993) applied the 2D vertical averaged k-ε turbulence model in an oxidation ditch simulation. In this paper, the authors used the 3D k-ε turbulence model to simulate the flow field in a small-scale operational ditch, and the simulated results were found to be acceptable by comparing with measured data. Based on the numerical simulation, detailed discussion follows and a possibly improved approach is suggested.

1 Governing equations

1.1 Flow field equations

The Reynolds-averaged, Navier-Stokes equations, governing the three-dimensional, steady, incompressible flow in the combined oxidation ditch as shown in Fig.1, are as follows:

Continuity equation:

$$\frac{\partial U_i}{\partial X_j} = 0. \tag{1}$$

Momentum equations:

$$\frac{\partial}{\partial X_j}(U_i U_j) = -\frac{1}{\rho}\frac{\partial P}{\partial X_i} + \frac{\partial}{\partial X_j}\left[v_t\left[\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right]\right], \tag{2}$$

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, v_t is the eddy viscosity, and ρ is the density of water.

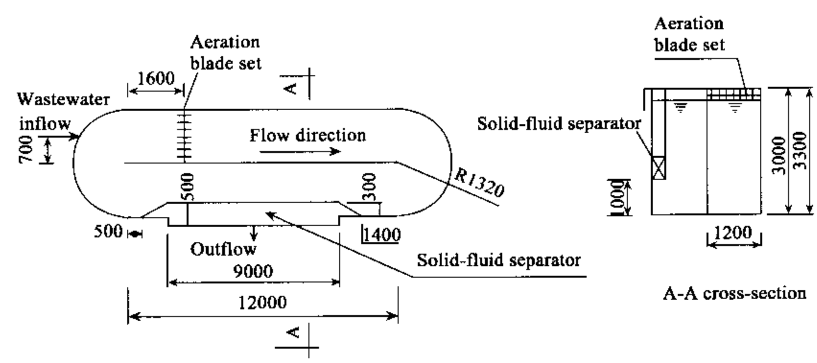

Fig.1 Diagram of the computed oxidation ditch (unit: mm)

1.2 k-ε turbulence model

The v_t in the above equations is determined with the k-ε turbulence model(Rodi, 1980):

$$v_t = C_\mu \frac{k^2}{\varepsilon}, \tag{3}$$

where the v_t is related by the turbulent kinetic energy, k, and the rate of its dissipation, ε.

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

k equation:

Foundation item: The Specialized Research Fund for the Doctoral Program of Higher Education(No. 20010610023) and the Sino-Finnish Scientific and Technological Cooperation Program

$$U_i \frac{\partial k}{\partial X_i} = \frac{\partial}{\partial X_i}\left[\frac{v_t}{\sigma_k}\frac{\partial k}{\partial X_i}\right] + G - \varepsilon. \qquad (4)$$

ε equation:

$$U_i \frac{\partial \varepsilon}{\partial X_i} = \frac{\partial}{\partial X_i}\left[\frac{v_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial X_i}\right] + C_{1\varepsilon}\frac{\varepsilon}{k}G - C_{2\varepsilon}\frac{\varepsilon^2}{k}, \qquad (5)$$

where G is the production term of turbulent energy by the mean velocity gradients:

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

The standard values of the k-ε turbulence model constants are used(Launder, 1972) :

$$C_\mu = 0.09,\ C_{1\varepsilon} = 1.44,\ C_{2\varepsilon} = 1.92,$$
$$\sigma_k = 1.0,\ \text{and}\ \sigma_\varepsilon = 1.3.$$

2 Computational domain

The computed case is a small-scale operational combined oxidation ditch. It has a volume of 94 m^3, a width of 1.2 m and depth of 3 m; the straight part of both sides is 12 m long, and the radius of the semi-circle part is 1.32 m. The central wall is 0.25 m in width with 0.125 m radius semi-circle ends. The diagram of the ditch is shown in Fig. 1. Since the wastewater flow rate(300 t/d = 0.00347 m^3/s) is relatively small compared with the average design velocity in the ditch (0.3 m/s), the inflow and outflow were not counted in the computation. Regardless, Stamou (Stamou, 1993) found that the inflow and outflow had little effect on the flow field in the ditch. A mechanical rotor was the only energy source to drag the ditch flow. It 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 groups of rotor blades along the rotating shaft, and 12 blades for each group. The angle between blades in the same group is 30 degree, and the adjacent groups are staggered, or at 15 degree angles with its neighbors. The shaft axis is 0.2 m above the water surface, and the blade's radius is 0.5 m and its width is 0.1 m. The deepest submerged part of a blade is 0.3 m. For a given computed moment, there are 4 and 5 blades submerged in liquid for every 3 other groups of blades. The angular velocity of the blades is 62 r/min.

Stamou (Stamou, 1993) used the x-directional momentum component to model the rotor power induced in his 2D simulation. In this study, 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. As shown in Fig.2, the part-cylindrical moving zone is 0.14 m in width, i.e. 0.02 m spaces between the side edge of the blades and the sidewalls of the zone on both sides. The outer wall of the zone is 0.04 m from the blade tips. Each zone is 0.04 m apart, and the distance between the first and the last zones and the side- and the central-walls of the ditch are 0.08 m. The sidewalls, the outer wall, and the surface of each zone form a complete moving zone. 6 moving zones are needed for 6 groups of blades. The sidewalls and the outer wall serve as the interfaces between the moving and the fixed mesh, which are the main sources of the drag force. Blades are built as baffle faces in the zone, or the blades are zero-depth solid surfaces in the computational domain(GRIDGEN user manual version 15, Pointwise, Inc., 2003). The blade surfaces are considered as moving walls, and the non-slip condition was set for them.

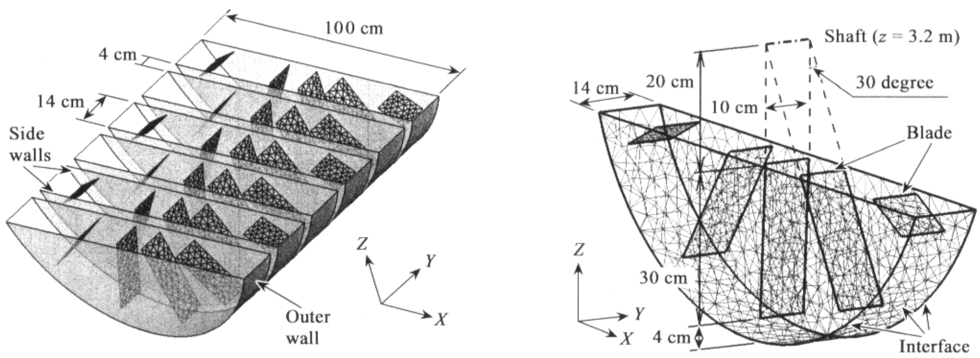

Fig.2 Blade and moving mesh setup

The bottom surface of the solid-liquid separator(Fig.1) was considered as the computational domain boundary, and all of the other parts of the separator were ignored.

The moving meshes and the fixed mesh of the ditch compose the computational domain. The whole meshes are unstructured, totaling 660826 tetrahedral cells. The averaged cell volume is 1.46×10^{-4} m^3, and the averaged tetrahedral edge length is 0.107 m. The grid system was built by the commercial CFD grid generation package, GRIDGEN.

3 Boundary conditions

The slip wall boundary condition was applied at the surface of the water, and the rigid-lid assumption was adopted. The bottom surface of the solid-liquid separator is set as a slip wall condition. For all of the walls, the non-slip boundary condition was supplied, including the bed, the side- and central-walls of the ditch, and the blade surfaces. The interface conditions were given between the moving and fixed meshes.

In order to simplify the computation, and refer to the Stamou's work(Stamou, 1993), neither inflow nor outflow was considered in this case.

4 Software package

The FLUENT package was chosen for implementing this case study. Segregated solver and the control-volume-based discretization scheme were used. All of the parameter settings held their default values in FLUENT, as shown in Table 1.

With a PC equipped with a Pentium III 500 MHz CPU, the total CPU time for a converged solution was about 24 h, or 1400 iteration steps.

Table 1 Parameters setup in FLUENT

Under-relaxation factors		Discretization	
Equation	Value	Equation	Scheme
Pressure	0.3	Pressure	Standard
Density	1.0	Pressure-velocity coupling	SIMPLE
Body forces	1.0	Momentum	First order upwind
Momentum	0.7	Turbulence kinetic energy	First order upwind
Turbulence kinetic energy	0.8	Turbulence dissipation rate	First order upwind
Turbulence dissipation rate	0.8		
Turbulence viscosity	1.0		

5 Results and discussion

Measurements were carried out in the computed ditch. A rotating fan velocimeter was used in the measurements. The velocity magnitude was probed at several sampling points, located at three different layers, the higher layer 0.4 m below water surface, the middle layer 1.5 m below water surface, and the lower layer 0.4 m from the ditch bed. The sampling points at every cross-section were located at the one- and two-third positions of the ditch width. The location details and numbering are shown in Fig.3.

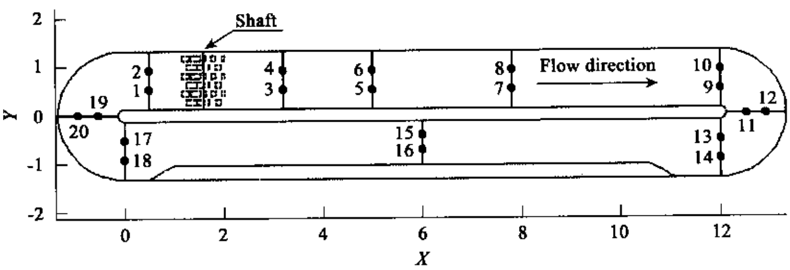

Fig.3 Sampling point locations

Fig.4 presents the computational velocity profiles and the measured data. As seen in the figures, most of the computational results matches well with the measured data. For those discrepant points, such as part points on line 1, 2, 11, 12 and 14, they were located near eddy areas. That means the velocity probing could be difficult and uncertain. Hence, the match between the measured data and the computational results is acceptable and good enough to support the numerical modeling and the assumption made earlier.

In order to get the whole picture of the flow phenomenon in the ditch, Fig.5(a—c) plots the velocity magnitude, the velocity vector projection on the horizontal planes and the vertical velocity contours at the three sampling layers, respectively. Several vortex areas were found: two right after each bend, which agrees with Stamou's 2D simulation (Stamou, 1993), and one at the lower part of the blade side near $x = 8$ m position. The existing recirculation can affect the performance of the ditch seriously. For a 3D understanding of the ditch flow, the X velocity contours at different cross-sections are shown in Fig.6. Combining Fig.5 and 6, they are seen clearly that a vertical reverse flow zone appears right before the far end bend from the blade, which could not be simulated in a 2D situation.

It should be noted that the velocity values were less than 0.3 m/s in a lot of areas, which is the design fluid velocity to prevent settlement of the suspended solid (SS). In addition to the reverse flow zones, the small velocity magnitude values appeared at the lower part on the rotor side and the higher part on the opposite side. The slow flow on the lower part of the rotor side came from the push of the vertical reverse flow mentioned above. Meanwhile, the vortex dragged the upper strong flow into the lower part of the ditch, resulting in the higher speed flow on the other side of it. After the second bend, or the bend before the rotor, the blades dragged the flow up and accelerated it again. Due to the flow pattern described here, the outflow zone, solid-liquid separator intake could be the heaviest SS laden part in the ditch, and the slow flow in the ditch could be the inducement of the SS settlement. In fact, the sedimentation was observed in the areas mentioned above, especially on the bed right under the blade, during the operation of the ditch. The authors believe that the main reason of SS settlement is the existence of vertical reverse flow zone and the low speed in lower part of the rotor side. The floating SS in the outflow was found during the operation, too. The higher velocity near solid-liquid separator intake should be the reason: SS was flushed out by rapid flow.

From Fig.5a, the velocity magnitude is sensitive to the inflow position. In other words, a little difference of the inflow position could result in big different flow track in the ditch, or the detention time of inflow could be quite differen under small different inflow position. According to the results above, suggestion can be made to improve the performance o a combined oxidation ditch:

In order to prevent settlement of SS in the ditch, the low speed zone at the shaft side should be avoided. The measures may include lifting the bottom, lowering the rotor, and/o changing the plane layout of the ditch.

The vertical reverse flow zone at the rotor side is a serious impact factor to the performance, and should b eliminated. The measures mentioned above can be use here, too. The outflow zone, or the intake zone of th separator, can be re-arranged to obtain better separatio efficiency.

The inflow positioning could be used to control detentio time of wastewater in the ditch. In this way the different loa of inflow can be handled.

It is surely that flow phenomena in an oxidation ditc have strong 3D features, especially for those single roto setups. It means that plug-flow assumption in a design, which is usually the case for designers which is not true. Therefore, either the current design criteria have limitation under this situation, or the function zones in a ditch could not be classified well, so the operation performance of the ditch could not be fully fulfilled.

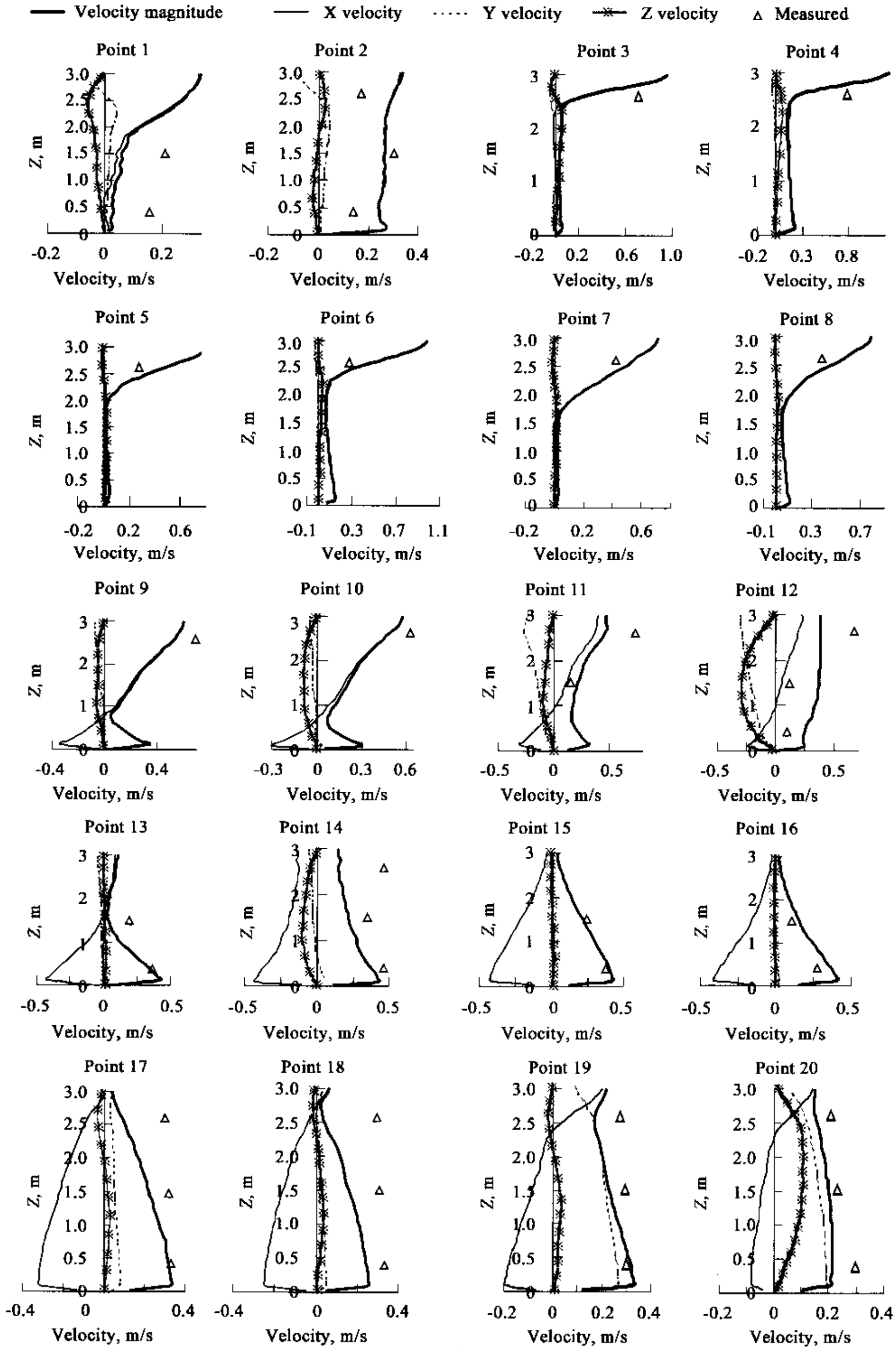

Fig.4 Calculated velocity profiles and measured data

a: Point 1 and Point 2; b: Point 3 and Point 4; c: Point 5 and Point 6; d: Point 7 and Point 8; e: Point 9 and Point 10; f: Point 11 and Point 12; g: Point 13 and Point 14; h: Point 15 and Point 16; i: Point 17 and Point 18; j: Point 19 and Point 20

6 Conclusions

A small-scale combined oxidation ditch flow was simulated by the 3D standard k-ε turbulence model, and the computed result was found to be acceptable by comparing it with the measured data.

A vertical reverse flow zone was found in the simulation, which affected the flow pattern greatly.

The slow flow areas appeared at the lower part of rotor side and the higher part of the other side, which are not suitable for an ideal ditch flow.

The outflow intake could be arranged on a higher position to improve the solid-liquid separation efficiency.

3D simulation should be a good supplement for improving the hydrodynamic performance in oxidation ditch designs.

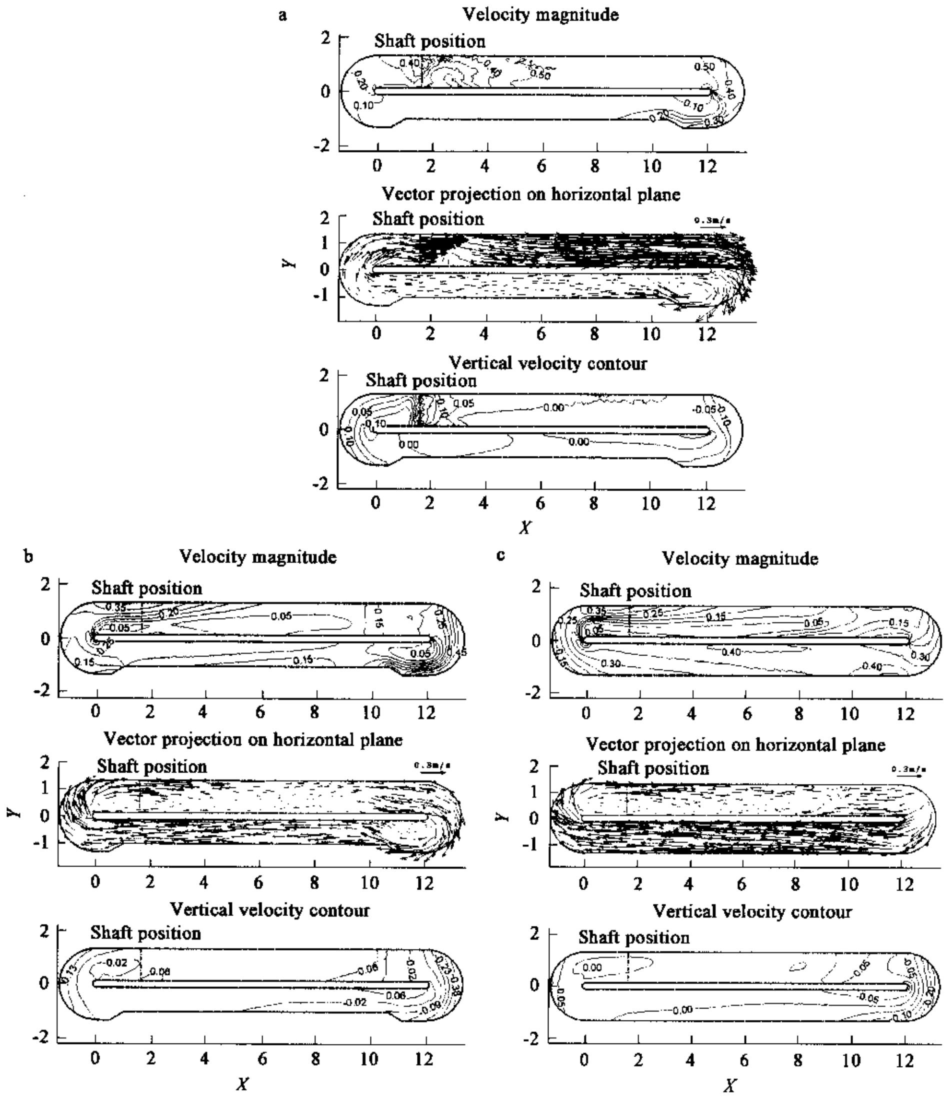

Fig.5 Velocity distribution at (a): the higher layer($z = 2.6$ m); (b): the middle layer($z = 1.5$ m); (c): the lower layer($z = 0.4$ m)

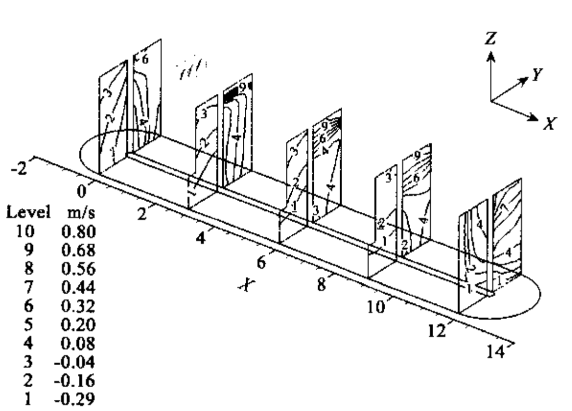

Fig.6 *X* velocity contour

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.

Accuracy laser doppler velocimeter(LDV) measurement for a model ditch is being proposed, by which researchers can explore the flow phenomenon and validate the numerical simulation in much more detail . It is hoped that numerical simulation can provide a powerful tool for future oxidation ditch designs.

References:

De Clercq B, Coen F, Vanderhaegen B *et al*., 1999. Calibrating simple model for mixing and flow propagation in waste water treatment plants[J]. Water Science and Technology, 39(4): 61—69.

Launder B E, Spalding D B, 1972. Lectures in mathematical models of turbulence [M]. England: Academic Press.

Lesage N, Sperandio M, Lafforgue C *et al*., 2003. Calibration and application of a 1-D model for oxidation ditches[J]. Chemical Engineering Research & Design, 81(A9): 1259-1264.

Rodi W, 1980. Prediction method for turbulent flows[M]. New York, USA: McGraw-Hill International Book Company.

Stamou A I, 1993. Prediction of hydrodynamic characteristics of oxidation ditches using the k-ε turbulence model [C]. 2nd Int. Symposium on Eng. Turbulence Modeling and Measurements, Florence, Italy. 261-271.

Stamou A I, 1994. Modeling oxidation ditches using the IAWPRC activated sludge model with hydrodynamic effects[J]. Water Science and Technology, 30 (2): 185-192.

Stamou A I, 1997. Modeling of oxidation ditches using an open channel flow 1-D advection-dispersion equation and ASM1 process description [J]. Water Science and Technology, 36(5): 269-276.

(Received for review December 10, 2004. Accepted February 23, 2005)