FLUCOME 2009
10th International Conference on Fluid Control, Measurements, and Visualization
August 17-21, 2009, Moscow, Russia

TWO-DIMENSIONAL NUMERICAL SIMULATION OF FLOW AND HEAT TRANSPORT

Hongwei Fang[1], Guojian He[1], Minghong Chen[2], Xiaobo Liu[3]

ABSTRACT

A depth averaged two-dimensional flow-pollutants coupled model is developed to evaluate impacts of industrial effluents on water environment with the framework of finite volume method (FVM) on an unstructured grid. The model employs Osher-type approximate Riemann solver to estimate the numerical flux of variables across the interface between cells and transfers 2-D problem into solving a series of local 1-D problem. The model is applied to calculate the hydrodynamic field of a Bay. The result agrees well with the measurements and represents the wet area and dry area changing under the tidal flow successfully. Further more, the model is used to forecast the diffusion area of thermal discharge with the tidal current from a power plant.

Key words: 2D numerical simulation, flow and heat transport, finite volume method, unstructured grid, tidal current

INTRODUCTION

The shallow-water equations are typically used to model river hydrodynamics and estuarine and coastal circulation. Many of these applications involve moving boundary and uneven bottom topography, and simulating these processes is becoming increasingly important. For regular geometries with simple boundaries, good results can be obtained with current numerical techniques (Chen, 1986). The classical method to treat irregular geometries in Cartesian coordinates is to use additional nodes from the intersection of the irregular geometry boundary and the Cartesian coordinates (Lapidus and Pinder, 1982). An improved method of computing flow field involving irregular geometries using a Cartesian grid was proposed by Chen et al. (1993) and is further developed by Fang and Liu (2006).

The application of finite volume method (FVM) to the numerical modeling of shallow water flows began in the 1980s and it was further developed in the late 1990s. The FVM has advantages over the finite difference method (FDM) and the finite element method (FEM), because FVM is suit for unstructured grid and synthesizes the simplicity and high efficiency of FDM and the geometry ability of FEM. Moreover, FVM solves the integral form of the conservation equations which can maintain the balance of the amount of mass and momentum. This method does not demand a continuous computational domain.

The reason of adopting unstructured grids is that these grids have more ability to fit the boundary and underwater geometry. The boundary of natural water and underwater geometry is so irregular that satisfied

[1] Department of Hydraulic Engineering, Tsinghua University; State Key Laboratory of Hydroscience and Engineering; Beijing 100084, China, fanghw@tsinghua.edu.cn
[2] College of Water Conservancy and Civil Engineering, China Agricultural University, Beijing 100083, China
[3] Environmental School, Beijing Normal University, Beijing 100875, China
Department of Water Environment, China Institute of Water Resources and Hydropower Research, Beijing 100038, China

structured grid is hard to get even though the desired level is not high. Unstructured gird is adopted in computational fluid dynamics more and more in recent years. They have universality in the fitness of computational domain.

In estuaries, the boundary of computational domain dynamically changes with the fluctuation of tide. The model must have an ability to capture this dynamic boundary while simulating quantum in these areas. FVM can be used for the calculation of tidal currents Alcrudo and Garcia-Navarro 1993 Balzano 1998 . The modeling of tidal currents has such unique features as a large computational domain complicated terrain conditions and complex open boundary conditions. The finite volume method has been used to discretize the governing equations and the space and time integral schemes are always the first-order schemes Wang and Zhang 1997 Xia 2002 . The regular rectangular grids are usually used in the published literature. With the development of calculating schemes of FVM and the methods of grid generation many new numerical models of FVM have been introduced into the simulation of tidal currents.

GOVERNING EQUATIONS

The conservation form of continuity equation and momentum equation for two-dimensional shallow water flows represents,

$$\frac{\partial q}{\partial t}+\frac{\partial f(q)}{\partial x}+\frac{\partial g(q)}{\partial y}=b(q) \tag{1}$$

where, $q=[h,hu,hv,hC]$ is a conservative quantum;

$f(q)=\left[hu,hu^2+gh^2/2,huv,huC\right]^T$;

$g(q)=\left[hv,huv,hv^2+gh^2/2,hvC\right]^T$;

$b(q)=\left[0,gh\left(S_{0x}-S_{fx}\right),gh\left(S_{0y}-S_{fy}\right),\nabla\cdot\left(D_i\nabla(hC)\right)+S/A-K\cdot hC\right]^T$;

$f(q)$ is flux in x direction; $g(q)$ is flux in y direction; $b(q)$ is a source term.

in which, h is water depth; u and v are depth-averaged horizontal velocity part in x and y directions, respectively; g is gravity acceleration; $S_{0x}=-\frac{\partial Z_b}{\partial x}$, which is the seabed slope in x direction; $S_{0y}=-\frac{\partial Z_b}{\partial y}$, which is the seabed slope in y direction. $S_{fx}=\frac{\rho u\sqrt{u^2+v^2}}{hc^2}=\frac{\rho n^2 u\sqrt{u^2+v^2}}{h^{4/3}}$, which is the resistant slope in x direction; $S_{fy}=\frac{\rho v\sqrt{u^2+v^2}}{hc^2}=\frac{\rho n^2 v\sqrt{u^2+v^2}}{h^{4/3}}$, which is the resistant slope in y direction.

DISCRETIZATION OF EQUATIONS

In irregular computational domain, Eq. (1) is solved by using finite volume method based on unstructured grid. If a matrix is defined as $F(q)=\left[f(q),g(q)\right]^T$ and integration on an arbitrary shaped element is evaluated for Eq. (1), one can get the basic formula for FVM,

$$\iint_\Omega q_t d\omega=-\int_{\partial\Omega}F(q)\cdot \mathrm{n}dl+\iint_\Omega b(q)d\omega \tag{2}$$

where, n is the unit outward normal vector of element boundary; $d\omega$ and dl are surface integral and linear integral, respectively. $F(q)\cdot n$ is flux in n direction, signed as $f_n(q)$. For one-order scheme, q is assumed as a constant in each element. Thus, according to divergence theorem, Eq. (2) can be discretized as,

$$A\frac{dq}{dt} = -\sum_{j=1}^{m} F_n^j(q)L^j + A\cdot b(q) \tag{3}$$

where, L^j is the length of the *j*-th edge of a element. For an element having *m* edges, the first term in right hand can be written as a summary of *m* terms. Each term equals the product of the normal flux ($f_n(q)$) of a function to be integrated across each edge of an element and the edge length.

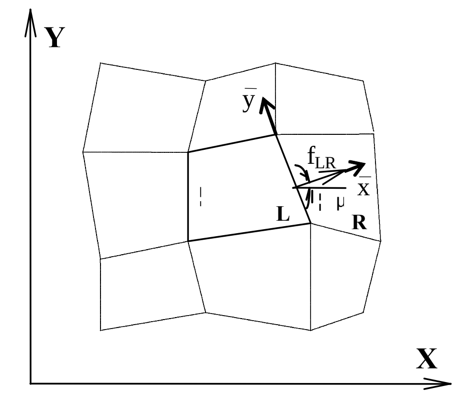

Fig. 1 A sketch of a finite volume Ω

If the angle between n direction and x axis is assumed to be Φ, which is shown in Fig. 1, one can get,

$$f_n(q) = \cos\Phi \cdot f(q) + \sin\Phi \cdot g(q) \tag{4}$$

According to the rotational invariance of function f and g, the following expressions are satisfied.

$$T(\Phi)f_n(q) = f[T(\Phi)q] = f(\bar{q}) \tag{5}$$

$$\text{or} \quad f_n(q) = T(\Phi)^{-1} f(\bar{q})$$

where, $T(\Phi)$ is a transformational matrix about rotational angle Φ, and $T(\Phi)^{-1}$ is a inverse transformational matrix.

$$T(\Phi) = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & \cos\Phi & -\sin\Phi & 0 \\ 0 & \sin\Phi & \cos\Phi & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix} \quad T(\Phi)^{-1} = T(\Phi)^T = \begin{bmatrix} 1 & 0 & 0 & 0 \\ 0 & \cos\Phi & \sin\Phi & 0 \\ 0 & -\sin\Phi & \cos\Phi & 0 \\ 0 & 0 & 0 & 1 \end{bmatrix}$$

The conservative quantum, vector q, transforms to $\bar{q} = T(\Phi)q$ through this transformational transformation, in which, $\bar{q}$ is a vector in the normal direction of an element edge. Through these projections and transformations, $g(q)$ disappears. So these two-dimensional problems changes into one-dimensional problems. The only work is to calculate $f(\bar{q})$. This makes the scheme more efficient.

Substituting Eq. (5) into Eq. (3), the final discretized FVM equation leads to,

$$A\frac{dq}{dt} = -\sum_{j=1}^{m} T(\phi)^{-1} f(\bar{q}) L^{j} + A \cdot b(q) \tag{6}$$

Because the value of q or $\bar{q}$ may not equal in both sides of one element edge, namely the value of q or $\bar{q}$ may has a discontinuity at the interface of a element, the solution of normal flux ($f(\bar{q})$) becomes a one-dimensional Riemann problem.

NORMAL NUMERICAL FLUX

Local one-dimensional Riemann problem is an initial-value problem. Its governing equation is,

$$\begin{aligned} &\frac{\partial \bar{q}}{\partial t} + \frac{\partial f(\bar{q})}{\partial \bar{x}} = 0 \\ &\bar{q} = \bar{q}_L \qquad \bar{x} < 0 \quad t = 0 \\ &\bar{q} = \bar{q}_R \qquad \bar{x} > 0 \quad t = 0 \end{aligned} \tag{7}$$

The origin of axis $\bar{x}$ is located at this edge and this axis points outward to normal direction. In Eq. (7), $\bar{q} = (h, h\bar{u}, h\bar{v}, hC)^T$; $\bar{q}_L$ and $\bar{q}_R$ are the value of vector $\bar{q}$ on left and right side of element edge. The expression of outward normal flux at origin when $t = 0^+$ under $\bar{x} - \bar{y}$ coordinate system can be attained by solving this Riemann problem. And it is assigned as $f_{LR}(\bar{q}_L, \bar{q}_R)$. Conducting an inverse rotational transformation on $f_{LR}(\bar{q}_L, \bar{q}_R)$, one can get the flux $f_n(q)$ across element edge under $x - y$ coordinate system.

BOUNDARIES

Above methodology is only suit for calculating normal numerical flux at the inner element edge within the computational domain. When the element edge is on the boundary of computational domain or object boundary such as hydraulic structure, flux calculation becomes to boundary Riemann questions. In this case, $\bar{q}_L$ is a known state in computational domain, while $\bar{q}_R$ is unknown. Generally, $\bar{q}_R$ is determined by choosing a suitable relationship between outputs in normal direction or specified boundary condition according to local fluid state.

Open boundary

(1)Given water elevation h_R

Deducing from $u_R + 2\sqrt{gh_R} = u_L + 2\sqrt{gh_L}$, one can get,

$$u_R = u_L + 2\sqrt{g}(\sqrt{h_L} - \sqrt{h_R}), \qquad v_R = v_L \tag{8}$$

(2)Given unit discharge Q_R

h_R and u_R can be obtained by solving equations

$$\begin{cases} Q_R = h_R u_R \\ u_R = u_L + 2\sqrt{g}(\sqrt{h_L} - \sqrt{h_R}) \end{cases} \tag{9}$$

where, $v_R = v_L$

(2)Given stage-discharge relation

If the stage-discharge relation $Q_R = f(h_R)$ at boundary is known, h_R and u_R can be achieved by solving equations

$$\begin{cases} Q_R = f(h_R) \\ u_R = u_L + 2\sqrt{g}(\sqrt{h_L} - \sqrt{h_R}) \end{cases} \tag{10}$$

where, $v_R = v_L$

Land boundary

A land boundary at the interface of an element represents that no current flows across this boundary. This state can be described as,

$$u_R = -u_L \qquad v_R = v_L \qquad h_R = h_L \tag{11}$$

TEMPERATURE DIFFUSIVITY EQUATIONS

The conservation equation governing movement of warm flows is,

$$\frac{\partial(h\Delta T)}{\partial t} + \frac{\partial(hu\Delta T)}{\partial x} + \frac{\partial(hv\Delta T)}{\partial y} = \frac{\partial}{\partial x}(D_{ix}h\frac{\partial \Delta T}{\partial x}) + \frac{\partial}{\partial y}(D_{iy}h\frac{\partial \Delta T}{\partial y}) - \frac{K\Delta T}{\rho C_P} + S_i \tag{12}$$

where, ΔT is temperature increment of water body; D_{ix} and D_{iy} are diffusion coefficients in x and y direction, respectively; K is a combined coefficient of heat transfer at water surface; C_P is the specific heat of water which is given as $1000cal/kg^{\circ}C$, ρ is water density; and S_i is source term.

a. Diffusion coefficient

$$D_i = 0.25\sqrt{ghi} \tag{13}$$

in which, i is water slope.

b. Combined coefficient of heat transfer at water surface

$$K = 15.7+(0.515\text{-}0.00425\times(T_s\text{-}T_0)+5.1\text{E-}5\times(T_s\text{-}T_0)^2)\times(70+0.7W_s\times 2) \tag{14}$$

in which, W_s is wind velocity; T_s is water temperature; T_0 is dew-point temperature.

CALCULATION EXAMPLE

The model developed above was applied to Dongshan Bay in Fujian, China. In order to simulate the flow dynamic field in computational domain accurately, calibration and validation are performed in this model. The calibration involves processes of tide line, flow velocity and flow direction in some typical spring tides, moderate tides and neap tides that are measured in plant boundary seas at 12 tide stations in 2008. The locations of 12 main stations in computational domain are shown in Fig.2.

The test period of time of spring tides, moderate tides and neap tides is listed as follows:

Spring tides: from 12:00 on July 5th, 2008 to 15:00 on July 6th

Moderate tides: from 10:00 on July 8th, 2008 to 12:00 on July 9th

Neap tides: from 7:00 on July 12th, 2008 to 10:00 on July 13th

The comparisons of tide line, flow velocity and flow direction between calculation and measurement are illustrated in Fig.3-5.

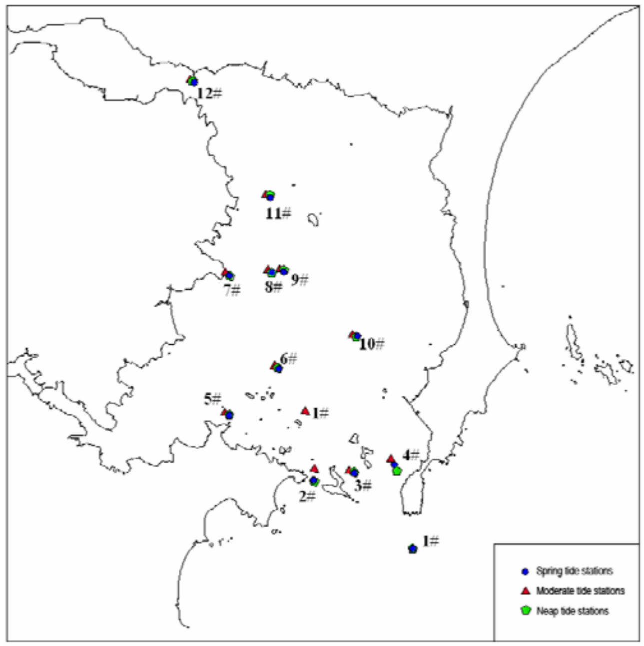

Fig.2 the locations of tide stations in computational domain

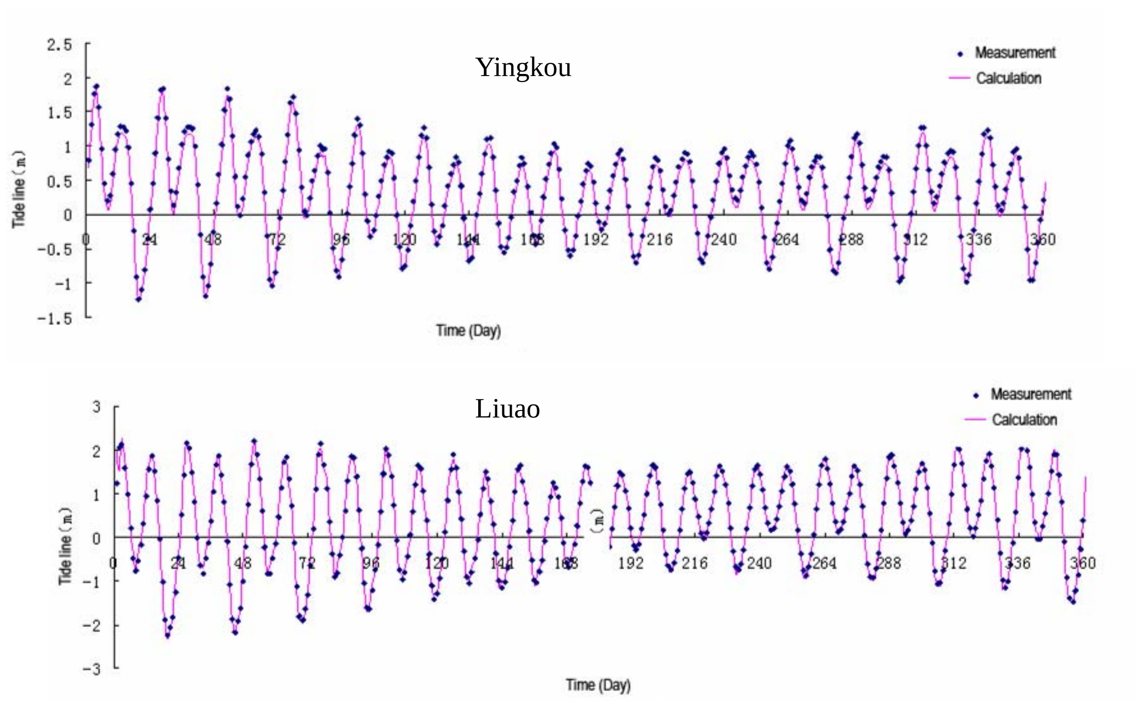

Fig.3 comparisons of tide line

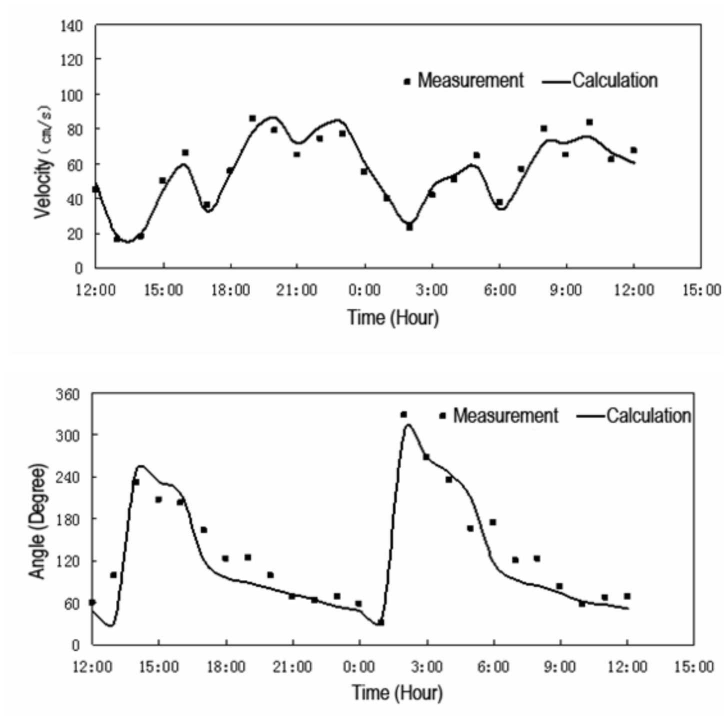

Fig.4 comparisons of flow velocity and flow direction under spring tide at 1#

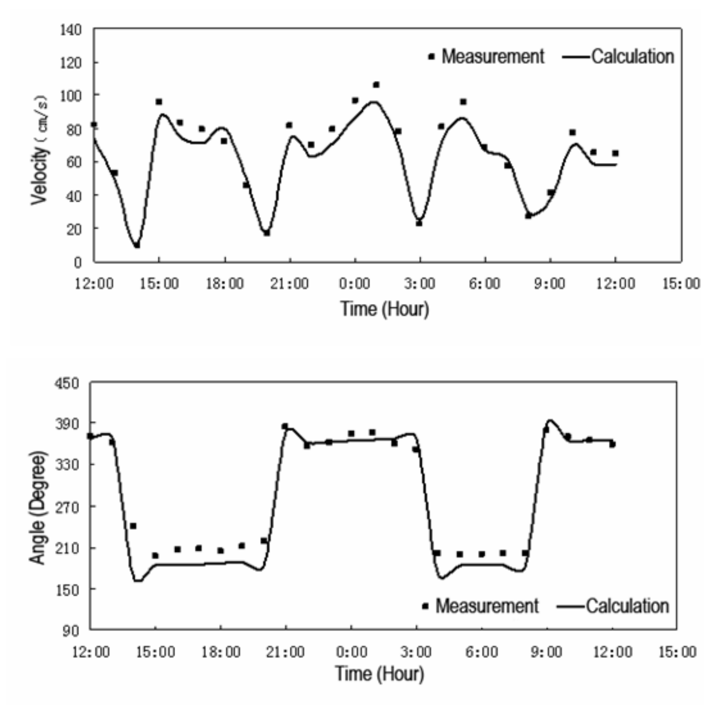

Fig.5 comparisons of flow velocity and flow direction under spring tide at 2#

The test model is applied to calculate the hydrodynamic field and heat transfer of the Bay. These simulation results including flow velocity field and temperatures of intake under various scenarios are illustrated in Fig. 6-8.

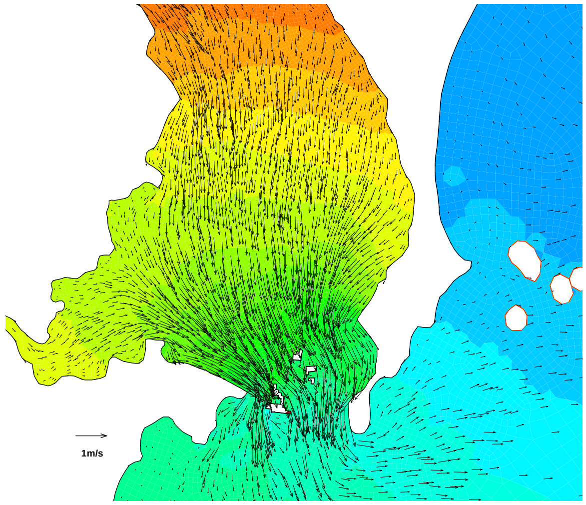

Fig.6 Calculated flow velocity field at 17:00 maximum ebb on July 5th

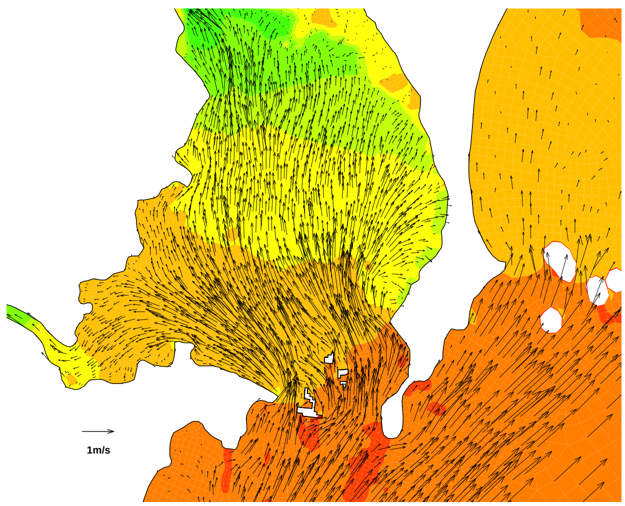

Fig.7 Calculated flow velocity field at 0:00 maximum rise on July 6th

It can be seen from the figures 6-8 that the character of calculated flow field agrees with the observation results from stations. The flow of outer sea forms reciprocating currents directing from southwest to northeast. Current flows into inner bay in spring tide. The main stream pools in channels on two sides with its average velocity beyond 1.0m/s. The velocity near intakes is also high. For heat transfer, the intake channel prevents the heat flow from dispersing toward north side.

CONCLUSIONS

This paper presents a depth averaged two-dimensional flow-pollutants coupled model. This model uses finite volume method with unstructured grid. The diffusion area of thermal discharge with the tidal current from a power plant in Zhangzhou is forecasted using this two-dimension model. The calibration for tide current field presents that the calculated tide line, fixed point flow velocity and flow direction agrees well with the original measurements and the flow state in computational domain squares with the fact observed. All these results show the capable of this model in simulating the flow dynamics and heat transfer in this region.

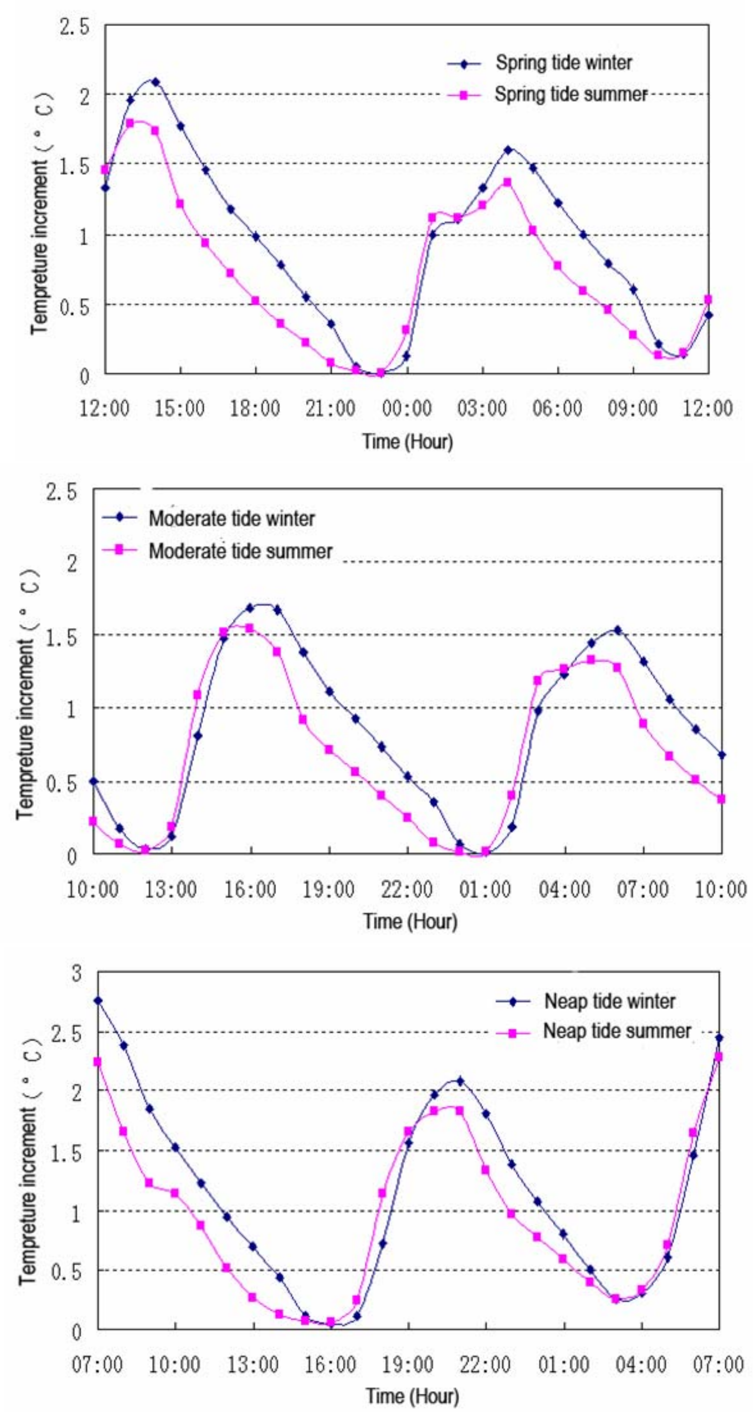

Fig.8 Temperature-time profile of intake under open channel scenario

REFERENCE:

[1] Alcrudo. F. and Garcia-Navarro P. A. High-Resolution Godunov-Type Scheme in Finite Volumes for the 2D Shallow-Water Equations, International Journal for Numerical Methods in Fluid, 199316, 489-505.

[2] Bradford, S. F., and Sanders, B. F. (2002). “Finite-volume Model for Shallow-water Flooding of Arbitrary Topograph.” J. Hydraul. Eng., ASCE, 128(3), 289-298.

[3] Carlson, K. D., Tien, H. C., and Chen, C. J. (1993). “Finite Analytic Numerical Simulation of Fluid Dynamics Problems Involving Irregular Geometries.” Proc., Refined Flow Modeling and Turbulence Measurements, Paris, 199-206.

[4] Chen, C. J., Tien, H. C., Carlson, K. D. and Bernatz, R. A. (1993). “Finite Analytic Method for 2D and 3D Flows with Complex Geometry”. Proc., Refined Flow Modeling and Turbulence Measurements, Paris, 161-174.

[5] Chen, C. J. (1986). “Finite Analytic Method.” Handbook of Numerical Heat Transfer, 723-746.

[6] Chen, C. J., Tien, H. C., Carlson, K. D. and Bernatz, R. A. (1993). “Finite Analytic Method for 2D and 3D Flows with Complex Geometry”. Proc., Refined Flow Modeling and Turbulence Measurements, Paris, 161-174.

[7] Fang HW, Liu B and Huang BB. Diagonal Cartesian Method for numerical simulation of flow and suspended sediment transport over complex boundaries. Journal of Hydraulic Engineering, ASCE, Vol.132, No.11, 2006, 1195-1205.

[8] Fang, H. W., Chen, C. J. and Lin, W. L. (2000). “Three-dimensional Diagonal Cartesian Method for Incompressible Flows involving Complex Boundaries.” Numerical Heat Transfer, B38(3), 37-57.

[9] Karpik, S. R., and Crockett, S. R. (1997). “Semi-lagrangian Algorithm for Two-dimensional Advection-Diffusion Equation on Curvilinear Meshes.” J. Hydraul. Eng., ASCE, 123(5), 389-401.

[10] Lin, W. L., Carlson, K. D., and Chen, C. J. (1998). “Diagonal Cartesian Method for Numerical Simulation of Incompressible Flow over Complex Boundary.” Numerical Heat Transfer, A33(5), 181-213.

[11] Lin, W. L., Carlson, K. D., and Chen, C. J. (1999). “Numerical Modeling of Conjugate Heat Transfer on Complex Geometries with Diagonal Cartesian Method -- Part I: Methods.” J. of Heat Transfer-transactions, ASME, 121(2), 253-260.

[12] Lapidus, L., and Pinder, G. F. (1982). “Numerical Solution of Partial Differential Equations in Science and Engineering.” Springger-verlag Inc., New York.

[13] Wang Xiaojian and Zhang Tingfang. Finite volume method for 3-D tide flow. Journal of Hydrodynamics, Ser. A, 1997, 12(1): 56-62.

[14] Xia Yunfeng. A simplified 3-D flow model for natural river with curvilinear collocated grids. Journal of Hydrodynamics, Ser. A, 2002, 17(5): 638 -646.