Proceedings of OMAE2006
25th International Conference on Offshore Mechanics and Arctic Engineering
June 4-9, 2006, Hamburg, Germany

OMAE2006-92424

A NUMERICAL SHALLOW WATER MODEL BASED ON THE NON-ORTHOGONAL CURVILINEAR GRIDS

Chaofeng Tong, Yanqiu Meng
State Key Laboratory of Hydrology-Water Resources and Hydraulic Engineering, Research Institute of Coastal & Ocean Engineering, Hohai University, 1 Xikang Road, Nanjing 210098, China
Email: tongchaofeng@126.com; Tel: +86 25 83786619, Fax: +86 25 83701905

ABSTRACT

According to the transformation relationships between the Cartesian coordinates and the general curvilinear coordinates, the governing equations of the model are derived as the forms in the general curvilinear coordinates from those in the Cartesian coordinates. In the model, the contravariant velocities are adopted as the independent variables in non-orthogonal grids. The momentum equations keep strongly conservative forms and the boundary conditions can be given easily. The model used a staggered grid arrangement. The discrete equations are solved using the SIMPLIC algorithms. The numerical model has been validated against the bifurcated flow of which the diversion angle is 30 degree. Compared with the measured values, the numerical shallow water model is shown to be capable of simulating the water domains with irregular boundaries.

INTRODUCTION

In the bifurcation channels or rivers including the estuaries, the orthogonal curvilinear grids are difficult to be generated in order to fit the irregular boundaries, especially when the diversion angle is not equal to 90 degree. It is inevitable to induce some inaccuracy when the orthogonal curvilinear coordinates are applied to the complex domain. Many numerical models have been developed to overcome this difficulty such as by use of the local mesh refinement based on Cartesian grid or orthogonal curvilinear grid and the finite element methods based on the triangle grids. An alternative approach is using general curvilinear grids whereby the meshes are generated to fit the boundaries correctly. Compared with the approaches based on the rectangular grid, the boundaries can be easier to be disposed in this approach. Since the contributions of non-orthogonal terms are considered, the accuracy can be improved. The equations and the approach are also simpler than those of the finite element methods requiring the inversion of the dense matrices. It can be said that the approaches based on the non-orthogonal curvilinear are very attractive.

There are three ways to select the main variables in solving the hydraulic equations based on the non-orthogonal grids, i.e. the Cartesian, covariant and contravariant velocity components are applied respectly. Ye et al. (1997), Karki and Mongia (1990), Shi et al.(1997) adopted the Cartesian velocity components as the main variables in the momentum equations. Xu and Zhang (1998) applied the covariant velocity components. Shi et al. (1998) used the contravariant velocity components. Because the contravariant velocity components are normal to the interfaces of grids, the fluxes across the grid interfaces and the boundary conditions can be conveniently expressed by use of the contravariant velocity components. On the other hand, there is still the cross pressure gradient term in each momentum equation of which the contravariant velocity components are adopted as the main variables as well as that uses the Cartesian velocity components. However, the cross term in every momentum equation is predominant along the direction of its covariant velocity component. It means that numerical methods on basis on the Cartesian systems can be applied to solve the equations based on the non-orthogonal grids systems. In the shallow water equations derived previously, such as reference (Shi et al.,1998) (Xie, 1999), which are based on the non-orthogonal Curvilinear coordinate systems and of which the contravariant velocity components are the main variables, the diffusion terms were neglected and the equations became simple. In fact, the neglects may increase errors and debase the stabilities or convergence of numerical solvers. So it is necessary to consider the diffusion terms in the shallow water equations.

It is the purpose of this paper to present details of a shallow water model for simulating the flow of bifurcation domains. First, the shallow water equations are transformed into the forms based on the non-orthogonal curvilinear coordinate systems with adopting the contravariant velocity components as the main variables. The hydrodynamic equations are discretized by the use of the finite volume method, and solved by SIMPLEC method. Finally, the model is applied to simulate and analyze the flow structure of bifurcation channels.

GOVERNING EQUATION BASED ON THE CARTESIAN COORDINATE SYSTEMS

In the shallow water domain, the flow is reasonably uniform and the values of different physical variables are almost constant in the vertical direction. The shallow water equations based on the Cartesian coordinate systems, also named 2-Dimensional hydrodynamic equations, are always expressed as

(1) The continuity equation

$$\frac{\partial \varsigma}{\partial t}+\frac{\partial Hu}{\partial x}+\frac{\partial Hv}{\partial y}=0 \tag{1}$$

(2) The momentum equations

$$\frac{\partial Hu}{\partial t}+\frac{\partial Hu^2}{\partial x}+\frac{\partial Hvu}{\partial y}=-gH\frac{\partial \varsigma}{\partial x}+2\frac{\partial}{\partial x}\left[H\nu_e\frac{\partial u}{\partial x}\right]+\frac{\partial}{\partial y}\left[H\nu_e\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\frac{\tau_{sx}}{\rho}-c_f u\sqrt{u^2+v^2}+fHv \tag{2}$$

$$\frac{\partial Hv}{\partial t}+\frac{\partial Huv}{\partial x}+\frac{\partial Hv^2}{\partial y}=-gH\frac{\partial \varsigma}{\partial y}+\frac{\partial}{\partial x}\left[H\nu_e\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+2\frac{\partial}{\partial y}\left[H\nu_e\frac{\partial v}{\partial y}\right]+\frac{\tau_{sy}}{\rho}-c_f v\sqrt{u^2+v^2}-fHu \tag{3}$$

where $H=\varsigma+h$ is the total water depth measured from the bed to the free surface, ς is the elevation of the free surface, h is the water depth measured from the bed to the static surface, t is time, u, v are the Cartesian velocity components, g is the gravity acceleration, $\nu_e=\nu_t+\nu$ is the general diffusion coefficient, here ν_t is the turbulent viscous coefficient and always is expressed as $\nu_t=C_\nu u_* H$ or $\nu_t=C_\nu u_*\Delta x$, $C_\nu=0.6\sim1.0$ is a constant, u_* is the friction velocity, Δx is the length of grid, ν is the kinemics viscous coefficient.

TRANSFORM OF THE GOVERNING EQUATIONS

In the Cartesian coordinate systems, the velocity vector $\vec{U}$ can be expressed as (u,v), which means the two velocity components along the X-direction and Y-direction respectively. In the general curvilinear coordinate systems, the velocity vector can also be written as (u_{con},v_{con}), u_{con} and v_{con} are the contravariant velocity components being normal to the η line and the ξ line respectively. The transformed relationship between these two kind coordinate systems can be derived as

$$u_{con}=\left(uy_\eta-vx_\eta\right)\Big/\sqrt{\alpha} \tag{4}$$

$$v_{con}=\left(ux_\xi-uy_\xi\right)\Big/\sqrt{\gamma} \tag{5}$$

where $\alpha=x_\eta^2+y_\eta^2$ is the length square in the η direction of the grid, $\gamma=x_\xi^2+y_\xi^2$ is the length square in the ξ direction of the grid.

In order to be expressed and compute conveniently, the contravariant velocity components always are written as the flux forms

$$U=\left(uy_\eta-vx_\eta\right) \tag{6}$$

$$V=\left(ux_\xi-uy_\xi\right) \tag{7}$$

So the Cartesian velocity components can also be represented by the contravariant velocity components as follows

$$u=\left(x_\xi U+x_\eta V\right)\Big/J \tag{8}$$

$$v=\left(y_\xi U+x_\eta V\right)\Big/J \tag{9}$$

where J is the Jacobian of the transformation.

By use of the transformation relationships of physical variables and their derivatives between the two coordinate systems, the shallow water equations in the general curvilinear coordinate system can be derived from in the Cartesian coordinate system. The governing equations including the continuity equation and the momentum equations can be written as the unite form.

$$\frac{\partial HJ\Phi}{\partial t}+\frac{\partial HU\Phi}{\partial \xi}+\frac{\partial HV\Phi}{\partial \eta}=\frac{\partial}{\partial \xi}\left[\frac{H\Gamma_\Phi}{J}\left(\alpha\frac{\partial\Phi}{\partial\xi}-\beta\frac{\partial\Phi}{\partial\eta}\right)\right]+\frac{\partial}{\partial \eta}\left[\frac{H\Gamma_\Phi}{J}\left(\gamma\frac{\partial\Phi}{\partial\eta}-\beta\frac{\partial\Phi}{\partial\xi}\right)\right]+S_\Phi \tag{10}$$

where $\Phi=(1,u,v)$ is denoted as the general dependent variable, $\beta=x_\xi x_\eta+y_\xi y_\eta$, S_Φ is the source term on the computation plane.

It is noted that these variables are still the Cartesian velocity components. The two momentum equations in the form of Eq.10 are disposed in order to get the momentum equations of which the main variables are the contravariant velocity components.

(1) $Eq.(10)(\Phi=u)\times y_\eta-Eq.(10)(\Phi=v)\times x_\eta$, the momentum equation of which U is the main variable can be derived.

$$\frac{\partial HJU}{\partial t}+\frac{\partial HU^2}{\partial \xi}+\frac{\partial HUV}{\partial \eta}=\frac{\partial}{\partial \xi}\left[\frac{H\nu_e}{J}\left(\alpha\frac{\partial U}{\partial \xi}-\beta\frac{\partial U}{\partial \eta}\right)\right]+\frac{\partial}{\partial \eta}\left[\frac{H\nu_e}{J}\left(\gamma\frac{\partial U}{\partial \eta}-\beta\frac{\partial U}{\partial \xi}\right)\right]+S_U \tag{11}$$

where, $S_U=S_{11}^U+S_{12}^U+S_{13}^U+S_{21}^U+S_{22}^U+S_3^U+P^U$,

$T_u=y_\eta v_\xi-y_\xi v_\eta$,

$T_v=x_\eta u_\xi-x_\xi u_\eta$,

$$S_{11}^U=\left[-\frac{H\nu_e u}{J}\left(\alpha y_{\xi\eta}-\beta y_{\eta\eta}\right)+\frac{H\nu_e v}{J}\left(\alpha x_{\xi\eta}-\beta x_{\eta\eta}\right)\right]_\xi+\left[-\frac{H\nu_e u}{J}\left(\gamma y_{\eta\eta}-\beta y_{\xi\eta}\right)+\frac{H\nu_e v}{J}\left(\gamma x_{\eta\eta}-\beta x_{\xi\eta}\right)\right]_\eta,$$

$$S_{12}^U=\left[-\frac{H\nu_e u_\xi}{J}\left(\alpha y_{\xi\eta}-\beta y_{\eta\eta}\right)+\frac{H\nu_e v_\xi}{J}\left(\alpha x_{\xi\eta}-\beta x_{\eta\eta}\right)\right]+\left[-\frac{H\nu_e u_\eta}{J}\left(\gamma y_{\eta\eta}-\beta y_{\xi\eta}\right)+\frac{H\nu_e v_\eta}{J}\left(\gamma x_{\eta\eta}-\beta x_{\xi\eta}\right)\right],$$

$$S_{13}^U=\left[\frac{H\nu_e}{J}\left(y_\eta^2 u_\xi-y_\xi y_\eta u_\eta-x_\eta T_u\right)\right]_\xi y_\eta+\left[\frac{H\nu_e}{J}\left(y_\xi^2 u_\eta-y_\xi y_\eta u_\xi+x_\xi T_u\right)\right]_\eta y_\eta-\left[\frac{H\nu_e}{J}\left(x_\eta^2 v_\xi-x_\xi x_\eta v_\eta-y_\eta T_v\right)\right]_\xi x_\eta-\left[\frac{H\nu_e}{J}\left(x_\xi^2 v_\eta-x_\xi x_\eta v_\xi+y_\xi T_v\right)\right]_\eta x_\eta,$$

$S_{21}^U=fH\left(\beta U+\alpha V\right)$,

$S_{22}^U=-C_f U\sqrt{\gamma U^2+\alpha V^2+2\beta UV}$,

$S_3^U=\left(-HU^2A_U-HV^2B_U-HUVC_U\right)/J$,

$A_U=x_{\xi\eta}y_\xi-y_{\xi\eta}x_\xi$, $B_U=x_{\eta\eta}y_\eta-y_{\eta\eta}x_\eta$,

$C_U=x_{\xi\eta}y_\eta-y_{\xi\eta}x_\eta+x_{\eta\eta}y_\xi-y_{\eta\eta}x_\xi$,

$P^U=-gH\left(\alpha\varsigma_\xi-\beta\varsigma_\eta\right)$.

(2) $Eq.(10)(\Phi=v)\times x_\xi-Eq.(10)(\Phi=u)\times y_\xi$, the momentum equation of which the V is the main variable can also be derived.

$$\frac{\partial HJV}{\partial t}+\frac{\partial HUV}{\partial \xi}+\frac{\partial HV^2}{\partial \eta}=\frac{\partial}{\partial \xi}\left[\frac{H\nu_e}{J}\left(\alpha\frac{\partial V}{\partial \xi}-\beta\frac{\partial V}{\partial \eta}\right)\right]+\frac{\partial}{\partial \eta}\left[\frac{H\nu_e}{J}\left(\gamma\frac{\partial V}{\partial \eta}-\beta\frac{\partial V}{\partial \xi}\right)\right]+S_V \tag{12}$$

where, $S_V=S_{11}^V+S_{12}^V+S_{13}^V+S_{21}^V+S_{22}^V+S_3^V+P^V$,

$$S_{11}^V=\left[-\frac{H\nu_e v}{J}\left(\alpha x_{\xi\xi}-\beta x_{\xi\eta}\right)+\frac{H\nu_e u}{J}\left(\alpha y_{\xi\xi}-\beta y_{\xi\eta}\right)\right]_\xi+\left[-\frac{H\nu_e v}{J}\left(\gamma x_{\xi\eta}-\beta x_{\xi\xi}\right)+\frac{H\nu_e u}{J}\left(\gamma y_{\xi\eta}-\beta y_{\xi\xi}\right)\right]_\eta,$$

$$S_{12}^V=\left[-\frac{H\nu_e v_\xi}{J}\left(\alpha x_{\xi\xi}-\beta x_{\xi\eta}\right)+\frac{H\nu_e u_\xi}{J}\left(\alpha y_{\xi\xi}-\beta y_{\xi\eta}\right)\right]+\left[-\frac{H\nu_e v_\eta}{J}\left(\gamma x_{\xi\eta}-\beta x_{\xi\xi}\right)+\frac{H\nu_e u_\eta}{J}\left(\gamma y_{\xi\eta}-\beta y_{\xi\xi}\right)\right],$$

$$S_{13}^V=-\left[\frac{H\nu_e}{J}\left(y_\eta^2 u_\xi-y_\xi y_\eta u_\eta-x_\eta T_u\right)\right]_\xi y_\xi-\left[\frac{H\nu_e}{J}\left(y_\xi^2 u_\eta-y_\xi y_\eta u_\xi+x_\xi T_u\right)\right]_\eta y_\xi+\left[\frac{H\nu_e}{J}\left(x_\eta^2 v_\xi-x_\xi x_\eta v_\eta-y_\eta T_v\right)\right]_\xi x_\xi+\left[\frac{H\varepsilon}{J}\left(x_\xi^2 v_\eta-x_\xi x_\eta v_\xi+y_\xi T_v\right)\right]_\eta x_\xi,$$

$S_{21}^V=-fH\left(\gamma U+\beta V\right)$,

$S_{22}^V=-C_f V\sqrt{\gamma U^2+\alpha V^2+2\beta UV}$,

$S_3^U=\left(-HU^2A_V-HV^2B_V-HUVC_V\right)/J$,

$A_V=y_{\xi\xi}x_\xi-x_{\xi\xi}y_\xi$, $B_V=y_{\xi\eta}x_\eta-x_{\xi\eta}y_\eta$,

$C_V=y_{\xi\xi}x_\eta-x_{\xi\xi}y_\eta+y_{\xi\eta}x_\xi-x_{\xi\eta}y_\xi$,

$P^V=-gH\left(\gamma\varsigma_\eta-\beta\varsigma_\xi\right)$.

It is showed that the transformed momentum equations with the contravariant velocity components as the independent variables in the non-orthogonal curvilinear systems keep the same forms in the Cartesian coordinate systems. Only the source term has been changed. The shallow water equations are still the conservation forms and the terms have the same physical meaning. Therefore the numerical methods used in the Cartesian coordinate systems can also be applied to solve the transformed shallow water equation.

BOUNDARY CONDITIONS

For the open boundaries, the water levels or velocities must be given. As a rule, only the discharges are offered in the upstream of rivers or estuaries. In this kind of condition, the boundary velocities along the river width can be obtained by the following equation.

$$\bar{u}_i(t) = H_i^{\frac{2}{3}}\left(\frac{B}{A}\right)^{\frac{2}{3}}\frac{Q(t)}{A} \tag{13}$$

where $\bar{u}_i(t)$ is the average velocity along water depth, A is the area of river cross section, B is the river width, the subscript i is denoted as the velocity location.

If the water levels are given in the boundaries of the computing domains, it is always assumed that the velocities on the boundary are satisfied with the expressions, $\frac{\partial U}{\partial \xi}=0$ (or $\frac{\partial V}{\partial \eta}=0$) and $V=0$ (or $U=0$). The water levels in the boundaries always can be gotten from the field measurement or the results of the numerical model applied to larger computation domain, also can be acquired after the harmonic analysis of tide.

The normal velocity is equal to zero at the lateral solid wall. The evaluation of tangential velocity should be dependent on the requirement of the model. When the numerical models are applied to the natural and large water domain, the non-slip conditions along the solid lateral walls may be adopted. For the small computation domains such as the channels, the boundary effects to the flow may be notable. So flow structures near the solid walls should be considered, the wall function always is used to compute the velocities and the drag force near solid walls, such as Eq.14 which is developed by Launder and Spalding (1972).

$$\frac{u}{u_*} = \frac{1}{\kappa}\ln\frac{z}{z_0} \tag{14}$$

where z_0 is the roughness height of solid wall, z is the distance from the velocity point to the wall. The Eq.14 can be applied to compute the friction velocity and the drag force coefficient.

In the rivers, coastal or estuary domains, the topographies are very irregular. The boundaries may vary due to the change of the water level. In order to solve the problems of the moving boundaries, many approaches are developed such as the dry and wet method, the line-boundary method, frozen boundary method et al.. The frozen boundary method is used in the model because it is simple.

NUMERICAL SCHEMES

The Finite Volume Method (FVM) is one of the most popular discretization methods in engineering CFD. For the outstanding mass conservation and easy treatment of boundary conditions for complex domains, it has been applied in some famous software, such as FLUENT. In the model, the method is used to discretize the equations. To avoid the pressure oscillations, staggered grids method or the momentum interpolation method of the collected grids can always be used. The method of staggered grids is simpler and applied in the model. The contravariant velocity fluxes are evaluated on the interfaces of the main grids, the water levels are located on the center of main grid cells.

The discrete equations of the shallow water equation can be rewritten as the following form

$$a_P\Phi_P = a_E\Phi_E + a_W\Phi_W + a_N\Phi_N + a_S\Phi_S + a^*\Phi^{old} + S_{\Phi C}\Delta\xi\Delta\eta + b_P^{\Phi} \tag{15}$$

where $a_E = D_e A(|P_{\Delta e}|) + \max(-F_e, 0)$,

$a_W = D_w A(|P_{\Delta w}|) + \max(F_w, 0)$,

$a_N = D_n A(|P_{\Delta n}|) + \max(-F_n, 0)$,

$a_S = D_s A(|P_{\Delta s}|) + \max(F_s, 0)$,

$D_e = \left(\frac{\alpha H\Gamma_\Phi}{J}\right)_e \frac{\Delta\eta_P}{\delta\xi_e}$, $D_w = \left(\frac{\alpha H\Gamma_\Phi}{J}\right)_w \frac{\Delta\eta_P}{\delta\xi_w}$,

$D_n = \left(\frac{\alpha H\Gamma_\Phi}{J}\right)_n \frac{\Delta\xi_P}{\delta\eta_n}$, $D_s = \left(\frac{\alpha H\Gamma_\Phi}{J}\right)_S \frac{\Delta\xi_P}{\delta\eta_s}$,

$F_e = (HU)_e\Delta\eta_P$, $F_w = (HU)_w\Delta\eta_P$,

$F_n = (HV)_n\Delta\xi_P$, $F_s = (HV)_s\Delta\xi_P$,

$P_{\Delta e} = F_e/D_e$, $P_{\Delta w} = F_w/D_w$,

$P_{\Delta n} = F_n/D_n$; $P_{\Delta s} = F_s/D_s$,

$A(|P|) = \max\left[0, (1-0.1|P|)^5\right]$,

$b_P^{\Phi} = \left(-\frac{H\Gamma_\Phi\beta}{J}\frac{\partial\Phi}{\partial\eta}\right)\Big|_w^e \Delta\eta + \left(-\frac{H\Gamma_\Phi\beta}{J}\frac{\partial\Phi}{\partial\xi}\right)\Big|_s^n \Delta\xi$,

$a^* = \frac{(HJ)_P^{old}\Delta\xi\Delta\eta}{\Delta t}$,

the superscript *old* denotes the time of $n\Delta t$. In order to improve the convergence of the numerical model , the source terms in the transformed governing equations are disposed as

$S_\Phi = S_{\Phi P}\Phi + S_{\Phi C}$, $S_{\Phi P} \le 0$,

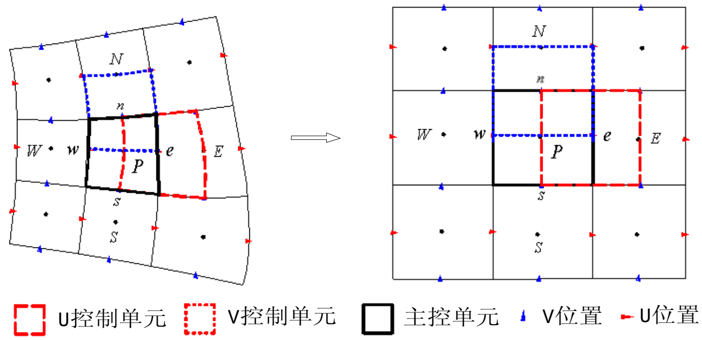
$a_P = a_E + a_W + a_N + a_S + a^* - S_{P\Phi}\Delta\xi\Delta\eta$,

U_e, U_w, V_n, V_s are the contravariant velocities at different interfaces of unit grid cell, $\Gamma_{\Phi e}, \Gamma_{\Phi w}, \Gamma_{\Phi n}, \Gamma_{\Phi s}$ are the diffusion coefficient at the grid cell interfaces, $\Delta\xi, \Delta\eta$ are the lengths of the grid cell in the directions of ξ, η, $\delta\xi_e, \delta\xi_w, \delta\eta_n, \delta\eta_s$ are the distances between every two grids next to each other.

Adopting the processes of the SIMPLEC methods to solve the discrete equations above in the model, the velocities and the water levels of computation domains can be acquired.

VALIDATION AND ANALYSIS

The validations of the model are supported and provided by the velocities measured in the bifurcated water channel. The model was designed to be consistent with one used in a set of experiments carried in the bifurcated channels (Tong, 2005).



U unit V unit Main unit Point V Point U

Physical plane Computational plane

Fig.1 Lay-out of grid systems

The bifurcated channels include one 0.4m wide main channel and one 0.3m wide branch channel, the angle of bifurcation is 30 degree, see Fig. 2. In the test, the velocities in the bifurcated domains were measured by 3D acoustic Doppler velocimeter (ADV). The fluxes of main channel downstream and branch channel are 0.0090884 m^3/s, 0.0027231 m^3/s respectively. The flux percent of the branch channel to the total upstream is 23.05%. In the diversion area, the average water depth is about 11.5cm. The locations of velocity to be measured are shown in Fig.3.

The number of grids applied in numerical computation is 280×41 including 134×41 for upstream of bifurcation point, 146×20 for the main channel and 146×20 for the branch channel. The grids are shown in Fig.3. The curvilinear grids on the left-up of the figure are the enlarged bifurcated region. The size of unit grids in that region is less than 2cm×2cm. In order to be consistent with the experimental data of the diversion flow ratios and the water levels at the bifurcated area, the water levels upstream are given as the boundary condition and the measured diversion fluxes are afforded as the downstream boundary condition. The water level of channel upstream is 0.12m and the roughness coefficient of organic glass n=0.010 is used to the bottom.

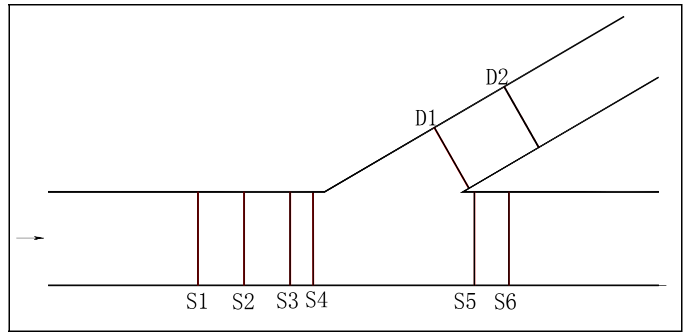


Fig.3 Locations of validation sections in bifurcated channels

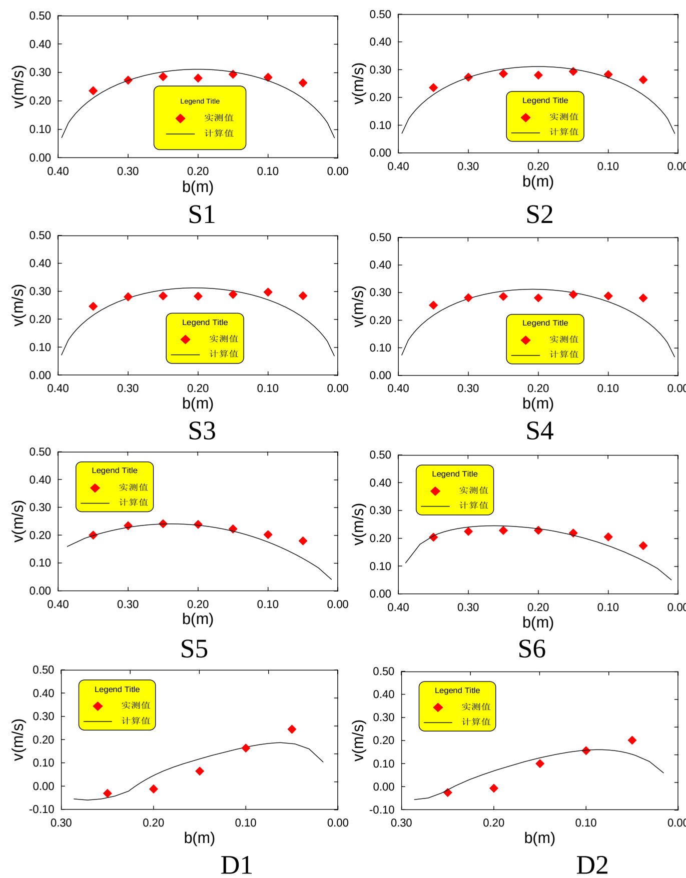


Fig.4 Comparisons of calculated velocities and measured velocities (dots are measured values, lines are computed values)

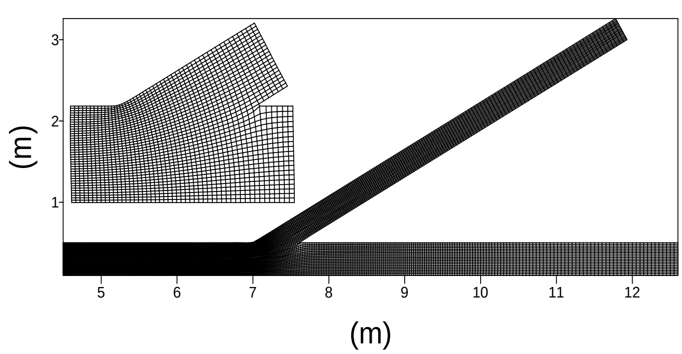


Fig.2 Computational grids of 30 degree bifurcated angle channels

The comparisons between the calculated velocities and the measured velocities in different cross sections are presented in Fig. 4. The dots represent the measured velocities and the lines are the computed values. It is shown that the velocities simulated by the model agree well with the experimental data only except that the deviations of velocities near the sold walls are a little bit larger. The results of the comparisons demonstrate that the numerical method can be applied to simulate the flow.

Fig. 5 shows the velocity contours of the bifurcated regions. It illustrates that the basic laws of the flow motion in the bifurcated regions, i.e. there are backflows existing at the leeside of the entrance to the branch channel that form the low velocity zone. At the same time, another low velocity area being symmetry with the above one is formed on the other side of the main channel. Furthermore, the distributions of the shear stresses and the erosion or deposition can be observed from the velocity contour graph.

The water level contours of 30 degree bifurcated angle channel are presented in Fig. 7. The distribution of water level contours of surface is similar to that of bend river. The water level on one side is higher than that on the opposite side. The reason is the same as the principles of the flow in the bend river, i.e. the flow bends when enters into the branch, then the transverse water level gradient occurs due to the difference of water level on both side in order to maintain the eccentric force. On the whole, the water surface on the side of main channel is higher than that on the side of the back flow zone of branch.

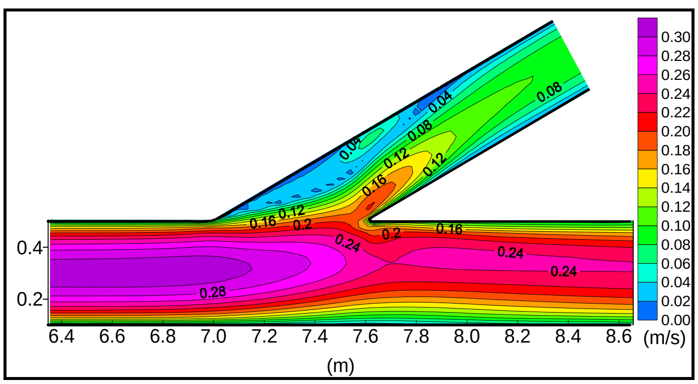


Fig.5 Velocity contours in bifurcated channel

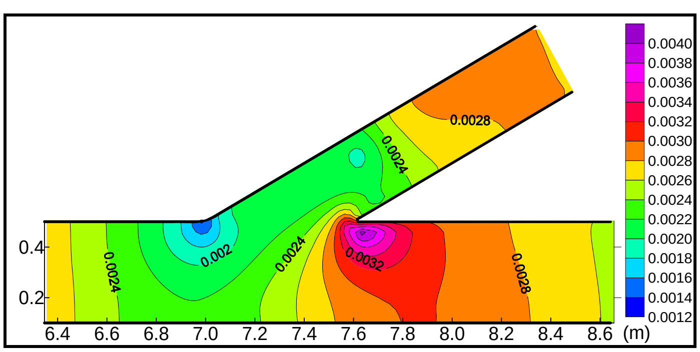


Fig.6 Water level contours in bifurcated channel

CONCLUSION

According to the transformation theories between the non-orthogonal curvilinear coordinate and the Cartesian coordinate, the hydrodynamic equations of shallow water with respect to the contravariant velocities as the main variables in the non-orthogonal curvilinear coordinate system are derived from the 2-D equations in the Cartesian coordinate systems. The effects of the non-orthogonal terms have been considered in the derived equations. The structures and conservative forms of the equations have been kept. With this advantage, it is easy to give the boundary conditions.

2-D flow numerical model has been established, in which the shallow water equations are discretized using FVM and are solved by the SIMPLEC method.. The numerical model has been validated against the bifurcated flow of which the diversion angle is 30 Degree. Compared with the measured values, the numerical shallow water model is shown to be capable of simulating the flow structures of the domain with irregular boundaries.

ACKNOWLEDGMENTS

This study is supported by National Natural Science Foundation of China (Grant No.50339010) and Open Research Fund of State Key Laboratory of Hydrology-Water Resources and Hydraulic Engineering of Hohai University (Grant No. 2005407211)

REFERENCES

[1] Jian Ye and J. A. McCorquodale, Depth-averaged hydrodynamic model in curvilinear collocated grid[J]. Journal of Hydraulic Engineering, ASCE, 1997,123(5): 380 -387.

[2] K.C.Karki and H.C.Mongia, Evaluation of a coupled solution approach for fluid flow calculations in body-fitted co-ordinates[J]. International Journal For Numerical Methods In Fluids, 1990,11:1-20.

[3] Fengyam Shi, Wenxin Sun, Gengsheng Wei. A WDM method on a generalized curvilinear grid for calculation of storm surge flooding[J]. Applied Ocean Research,1997,19: 275-282.

[4] H. Xu and C. Zhang, Study of the effect of the non-orthogality for non-staggered grids--the theory[J]. International Journal of Numerical Methods in Fluids, 1998,28:1265-1280.

[5] Fengyan Shi, Zhenya Kong and Xinping Ding. A implicit method using contravariant velocity components and its application to the calculations in a harbour-channel area[J]. Acta oceanologica sinica. 1998,20(4): 17-24. (in Chinese)

[6] Jun Xie. A 2-D tidal numerical model in the general grid coordinates systems[C]. The 9th conference of Coastal engineering of China. Beijing：Ocean Press.1999: 335-341．(in Chinese)

[7] Chaofeng Tong, Study on Flow and Sediment Movement of Bifurcation area and Related Application of 3-D Numerical Flow Model, Ph.D. thesis, University of Hohai,2005. (in Chinese)