A lattice Boltzmann model coupled with a Large Eddy Simulation model for flows around a groyne

Xin YU\textsuperscript{1}, Xuelin TANG\textsuperscript{2}, Wuchang WANG\textsuperscript{3}, Fujun WANG\textsuperscript{4}, Zhicong CHEN\textsuperscript{5}, and Xiaoyan SHI\textsuperscript{6}

Abstract
This present paper proposes a two-dimensional lattice Boltzmann model coupled with a Large Eddy Simulation (LES) model and applies it to flows around a non-submerged groyne in a channel. The LES of shallow water equations is efficiently performed using the Lattice Boltzmann Method (LBM) and the turbulence can be taken into account in conjunction with the Smagorinsky Sub-Grid Stress (SGS) model. The bounce-back scheme of the non-equilibrium part of the distribution function is used to determine the unknown distribution functions at inflow boundary, the zero gradient of the distribution function is set normal to outflow boundary to obtain the unknown distribution functions here and the bounce-back scheme, which states that an incoming particle towards the boundary is bounced back into fluid, is applied to the solid wall to ensure non-slip boundary conditions. The initial flow field is defined firstly and then is used to calculate the local equilibrium distributions as initial conditions of the distribution functions. These coupled models successfully predict the flow characteristics, such as circulating flow, velocity and water depth distributions. The comparisons between the simulated results and the experimental data show that the model scheme has the capacity to solve the complex flows in shallow water with reasonable accuracy and reliability.

Key Words: Two-dimensional flows, Shallow water equations, Lattice Boltzmann Method (LBM), Sub-Grid Stress (SGS), Non-submerged groyne

1 Introduction
Groynes are often used in river engineering for navigation improvement or bank support, as well as in coastal engineering for bank or beach protection. The groynes placed in a river result in flow constrictions and deflections with strongly circulating features. The probability of deposition of sediment and silt in a river reach as well as the silt retention time is strongly affected by the details of the flow field in stagnant zones and the intermediate mixing layer. It is necessary to clarify flow structures and instantaneous vortex structures around groynes. Experimentation is an essential and useful tool to investigate the flow around groynes (Uijttewaal, 2004).

In the last two decades, extensive research has been conducted to investigate various methods to solve shallow water equations (Wang et al., 2004). Two-dimensional (2D) depth-averaged equations have been

\textsuperscript{1} Doctoral student, College of Water Conservancy & Civil Engineering, China Agricultural University, Beijing 100083, China; Senior Engineer, Yellow River Institute of Hydraulic Research, Zhengzhou 450003, China.
\textsuperscript{2} Assoc. Prof., College of Water Conservancy & Civil Engineering, China Agricultural University, Beijing 100083, China. Corresponding author, E-mail: xl-tang@mail.tsinghua.edu.cn
\textsuperscript{3} Graduate Student, College of Water Conservancy & Civil Engineering, China Agricultural University, Beijing 100083, China.
\textsuperscript{4} Prof., College of Water Conservancy & Civil Engineering, China Agricultural University, Beijing 100083, China.
\textsuperscript{5} Prof., State Key Laboratory of Hydroscience and Engineering, Tsinghua University, Beijing 100084, China.
\textsuperscript{6} Assoc. Prof., College of Water Conservancy & Civil Engineering, China Agricultural University, Beijing 100083, China.

Note: The original manuscript of this paper was received in Feb. 2009. The revised version was received in April 2010. Discussion open until Sept. 2011.
obtained by integrating the Navier-Stokes equations from the channel bottom to the water surface to simplify the governing equations (Aliparast, 2009). With the rapid advances in computer technology, numerical techniques have become vital for a full understanding of the flows around groynes. Numerical model simulations are usually more cost-effective and faster than physical model studies, and have no inherent limitations on spatial extent. Tingsanchali and Maheswaran (1990) used a 2D depth-averaged model, incorporating a correction factor into the $k - e$ model and introducing a three-dimensional correction factor to improve the computed bottom stresses. Molls and Chaudhry (1995) further developed a general mathematical model to solve the unsteady 2D depth-averaged equations by combining a constant eddy-viscosity turbulence model. Moreover, Molls and et al. (1995) extended this model to numerical simulation of 2D flow near a groyne applying a constant eddy-viscosity coefficient obtained from experimental data to the entire domain, which depended on the estimate of eddy viscosity near the groyne. As a result of the rapid advancement of Computational Fluid Dynamics (CFD) in recent years, Large Eddy Simulation (LES) has become one of the most important and promising CFD techniques because it requires less computer time than Direct Numerical Simulation (DNS) and uses simpler and more universal models than simulations based on the Reynolds-averaged Navier-Stokes equation model (RANS). The LES is a turbulence simulation method, and assumes that the turbulent motion can be separated into large-scale (grid-scale) eddies and small-scale eddies. The separation between them does not have a significant effect on the evolution of the large eddies, and the small eddies are independent from the flow geometry and the boundary conditions. The LES solves the large-scale field directly by a set of filtered governing equations, while the SGS is isotropic and mainly serves to represent dissipation, so its effect on the grid scale can be easily modeled (Lesieur et al., 2005). Yue et al. (2006) established a large-eddy simulation code to model turbulent open-channel flow over a two-dimensional dune. Turbulence statistics and instantaneous flow structures are examined. Numerical results from two computational grids agree with each other, and are also in good agreement with recently-obtained experimental data. Details of the separated flow and development of the flow after reattachment are well predicted. Based on the rigid lid assumption, Tang et al. (2006) proposed a dynamic Smagorinsky model and numerically simulated the three-dimensional vortex flows around the groyne, and the computational results were in fairly good agreement with the experimental data, but the computational time was too large.

The Lattice Boltzmann Method (LBM) is successfully applied to the analysis of a variety of complex physical phenomena, such as turbulent flow (Yu et al., 2005) and advection and anisotropic dispersion (Zhang et al., 2002). Unlike conventional numerical methods based on discretization of macroscopic equations, LBM is based on a microscopic model and mesoscopic kinetic equations (Chen et al., 1998). Zhou (2002) developed a lattice Boltzmann model for the shallow water equations with a standard SGS model and solved two flow problems, such as flows in a straight channel and over a submerged island. The numerical predictions are verified by the analytical solutions and available experimental data. Djendi (2006) used the LBM to carry out a direct numerical simulation of grid-generated turbulence with the view to improving comparison between experimental and numerical results on approximate isotropic turbulence. The results compare relatively well with existing experimental data on grid turbulence. Liu et al. (2009) developed a two-dimensional LBM for sub-critical flows in open channel junctions coupled with the large eddy simulation model and took into account the turbulence. A multi-block lattice scheme is designed and applied to the area of combining flows. The simulated results are compared with available experimental data and classical analytical solutions. Ghidaoui (2008) presented an introduction of the main concepts of kinetic theory and a brief summary of some of its applications in hydraulics and fluid mechanics, and offered information about the potential of using the BGK framework in modeling systems with large interacting individuals such as sediments and eddies. Fernandino et al. (2009) predicted the turbulent open duct flow using the LBM in conjunction with the Smagorinsky SGS model. The results for the mean flow and turbulent fluctuations are in good qualitative agreement with the experiments. Based on the dynamic SGS model, Premnath et al. (2009) employed the multiple relaxation times (MRT) formulation of LBM to compute turbulent statistics such as the root-mean-square velocity fluctuations and Reynolds stresses, and the simulated results are in good agreement with prior DNS and experimental data.

The aim of the present study is to evaluate this simple LBM-SGS model as a possible tool for the simulation of 2-D shallow water flows, as a first step for the future application of this model to the study...
of the typical secondary flows around the groyne. For the sake of understanding the flow pattern around a groyne and verifying the effectiveness of the numerical model, a physical flume model with a groyne fixed along the flume side is constructed and these flows around the groyne are experimentally investigated. A lattice Boltzmann model coupled with a SGS model is proposed to model the flows around the groyne, and then much more flow information is also numerically predicted and the data are analyzed systematically. Finally, comparisons are drawn between the computational results and the experimental data.

2 Model development

2.1 LES governing equations

Three-dimensional (3D) turbulent flow is a quite common phenomenon in natural water flows and can be simulated using the Navier-Stokes equations and the continuity equation theoretically. Compared to the DNS and the RANS, the LES requires less computer resources than the DNS and uses simpler and more universal models than the RANS, thus has become an important and promising approach for engineering applications. Therefore, the large eddy simulation is used here to model turbulent flows and most SGS models are based on eddy-viscosity assumptions. The most commonly used model is the Smagorinsky Model (SM), and based on the assumption of the balance between the energy production and dissipation, the SGS is assumed to be proportional to the resolved strain-rate tensor. The 3D filtered governing equations can be written as (Tang, 2006)

\[
\frac{\partial \vec{u}}{\partial t} = 0
\]

\[
\frac{\partial \vec{u}}{\partial t} + \frac{\partial (\vec{u} \cdot \vec{u})}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + v_e \frac{\partial^2 \vec{u}}{\partial x_i \partial x_j} + f_i
\]

where \(i\) and \(j\) are indices and the Einstein summation convention is used, i.e. repeated indices mean a summation over the space coordinates; \(t\) is the time; \(x_1, x_2, x_3\) represents coordinate \(x, y, z\) respectively; \(\rho\) is the water density; \(p\) is the pressure; \(v_e\) is the effective viscosity coefficient including the molecular viscosity coefficient \(\nu\) and the turbulent eddy viscosity coefficient \(v_t\), i.e. \(v_e = \nu + v_t\); \(f_i\) is the body force per unit mass in the \(i\) direction; \(\vec{u}_i\) is the space-filtered component in the \(i\) direction defined by \(\vec{u}_i(x,y,z) = \int_{\xi_i}^{\xi_i+\Delta x_i} \int_{\xi_j}^{\xi_j+\Delta x_j} \int_{\xi_k}^{\xi_k+\Delta x_k} G(x, y, z) \vec{u}(x, y, z) d\xi_i d\xi_j d\xi_k\), \(\vec{u}_i\) is the velocity component before being space-filtered, \(G\) is the grid filter with filter width \(\Delta x\), \(\xi_i (i = 1, 2, 3)\) is the \(i\) integral variable.

However, 3D numerical simulations are too expensive in calculations to be widely used. Therefore, in order to simplify 3D governing equations, the assumption that the vertical acceleration is ignored is often made so that the hydrostatic pressure distribution replaces the momentum equation in the vertical direction in the mathematical model. Therefore, the 2D depth-averaged equations are obtained by integrating the filtered governing equations from the channel bottom to the water surface and describe the horizontal structure of shallow water flows. However, this type of method cannot provide the detailed structure of the turbulence and the information concerning the vertical velocity distribution is lost. In the case of small-channel bottom slope, negligible Coriolis acceleration and negligible wind shear, the 2D space-filtered shallow water governing equations can be written in the following tensor form (Zhou, 2002).

\[
\frac{\partial h}{\partial t} + \frac{\partial (h \vec{u}_i)}{\partial x_i} = 0
\]

\[
\frac{\partial (h \vec{u}_i)}{\partial t} + \frac{\partial (h \vec{u}_i \vec{u}_j)}{\partial x_j} = -\frac{g \partial h^2}{2} \frac{\partial h}{\partial x_i} + v_e \frac{\partial^2 \vec{u}_i}{\partial x_i \partial x_j} - \frac{\tau_{xi}}{\rho} - gh \frac{\partial z_s}{\partial x_i}
\]

where \(h\) is the water depth; the depth-averaged velocity component \(\vec{u}_i (i = 1, 2)\) in the \(i\) direction is defined as \(\vec{u}_i = \frac{1}{h} \int_{z_s}^{z_s+\Delta z} \vec{u}(z) dz\); \(z_s\) is the bed elevation above datum; \(g = 9.81 m/s^2\) is the gravitational acceleration.
acceleration; the bed shear stress $\tau_{bi}$ in the $i$ direction is written as $\tau_{bi} = \rho C_{b} \sqrt{u_{j} S_{ij}}$, $C_{b}$ is the bed friction coefficient.

The SM is employed here, thus the SGS $\tau_{ij}$ can be written as follows.

$$\tau_{ij} = -2\nu_{i} S_{ij}$$

where the SGS eddy viscosity coefficient $\nu_{i} = C_{s} \frac{\sqrt{S}}{S_{ij}}$, $C_{s}$ is the Smagorinsky constant varying from 0.1 to 0.27, $\sqrt{S}$ is the magnitude of $\sqrt{2 S_{ij} S_{ij}}$, $S_{ij}$ is the depth-averaged and filtered strain-rate tensor here defined by $S_{ij} = \frac{1}{2\Delta} \left( \frac{\partial (h U_{i})}{\partial x_j} + \frac{\partial (h U_{j})}{\partial x_i} \right)$.

2.2 Lattice Boltzmann model

According to the LBM theory, it has three main components: the kinetic equation, a lattice pattern, and the local equilibrium distributions. A lattice Boltzmann model consists of two steps: a streaming step and a collision step (Chen and Doolen, 1998). Based on Bhatnagar-Gross-Krook (BGK) approximation of the collision operator and a 9-speed square lattice shown in Fig. 1, and after incorporating the SGS model, the lattice Boltzmann equation in the $D$ direction can be written as

$$f_{\alpha}(x + e_{\alpha} \Delta t + \Delta \alpha) - f_{\alpha}(x,t) = \frac{1}{\tau_e} [f_{\alpha}(x,t) - f_{\alpha}^{eq}(x,t)] + \frac{\partial}{\partial x_{\alpha}} e_{\alpha} F_{\alpha}, \alpha = 0 - 8$$

where $f_{\alpha}$ is the particle distribution function in the $\alpha$ direction; $e_{\alpha}$ is the particle velocity vector in the $\alpha$ link; $\Delta t$ is the time step; $\tau_e$ is the total relaxation time including the single relaxation time $\tau$ with respect to the molecular viscosity $\nu$ and the eddy relaxation time $\tau_e$ associated with turbulent eddy viscosity $\nu_e$, i.e. $\tau_e = \tau + \tau_e$, which determines the approaching rate to the local equilibrium; $f_{\alpha}^{eq}$ is the local equilibrium distribution function; $e = e_{x}/\Delta x$, $\Delta x$ is the lattice size; $F_{\alpha}$ is the component of the force in the $\alpha$ direction defined as $F_{\alpha} = -\frac{e_{\alpha} p}{\rho} g \frac{\partial z}{\partial \alpha} - \frac{N_a}{\rho} e_{\alpha} e_{\alpha} \Delta \alpha$; $N_a$ is a constant determined by the lattice pattern as $N_a = \frac{1}{e_{\alpha}} \sum e_{\alpha} e_{\alpha} \Delta \alpha = 6$; $e_{i\alpha}$ is the component of a particle velocity vector $e_{\alpha}$ in the $i$ spatial coordinate.

For the 9-speed square lattice shown in Fig. 1, each particle streams one lattice unit at its velocity along one of the eight links indicated with indexes 1–8, and index 0 indicates the particle at rest with zero speed. The particle velocity vector $e_{\alpha}$ in the $\alpha$ link is defined by

Fig. 1 Lattice pattern: D2Q9
Applying the Chapman-Enskog expansion to the lattice Boltzmann equation can recover the 2D shallow water governing equations combined with the SGS turbulence model, and the local equilibrium distribution function is expressed as

\[
\begin{align*}
e_{a} = \begin{cases}
(0,0), & \alpha = 0 \\
\cos\left(\frac{(\alpha-1)\pi}{4}\right), \sin\left(\frac{(\alpha-1)\pi}{4}\right), & \alpha = 1, 3, 5, 6 \\
\sqrt{2\alpha}\cos\left(\frac{(\alpha-1)\pi}{4}\right), \sin\left(\frac{(\alpha-1)\pi}{4}\right), & \alpha = 2, 4, 6, 8
\end{cases}
\end{align*}
\]  

(7)

From the distribution function, the water depth \( h \) and flow velocity \( u_i \) can be calculated from the following summation formulas.

\[
h = \sum_{\alpha} f_{\alpha}u_{r} - \frac{1}{h} \sum_{\alpha} f_{\alpha}e_{\alpha} \tag{9}
\]

The relation between the effective viscosity coefficient and the total relaxation time can be expressed as

\[
v_{e} = \frac{\hat{\delta} t}{6} (2\tau - 1) \tag{10}
\]

At the same time, the molecular viscosity coefficient and the single relaxation time is assumed to satisfy the relation

\[
v = \frac{\hat{\delta} t}{6} (2\tau - 1) \tag{11}
\]

And then, we obtain the following relation between the eddy relaxation time and the turbulence viscosity.

\[
\tau_e = \frac{3v_{e}}{\hat{\delta} t} \tag{12}
\]

In order to obtain the total relaxation time \( \tau_e \), first of all, we need to determine \( \tau_e \) using the strain-rate tensor \( \Sigma_y \). In the lattice Boltzmann model, it is natural to calculate \( \Sigma_y \) in terms of the distribution function. By using the Chapman-Enskog expansion, it can be found that the strain-rate tensor \( \Sigma_y \) is related to the non-equilibrium momentum flux tensor expressed by

\[
\Sigma_y = \frac{1}{2h} \left[ \frac{\partial (\hat{h} \hat{\nu}_j)}{\partial x_j} + \frac{\partial (\hat{h} \hat{\nu}_r)}{\partial x_j} \right] = -\frac{3}{2e^2h\tau_e\delta t} \sum_{\alpha} e_{\alpha}e_{\alpha}(f_{\alpha} - f_{\alpha}^{eq}) \tag{13}
\]

Then the turbulent viscosity \( v_{e} \) can be rewritten as

\[
v_{e} = C_{\Lambda} \frac{\hat{\Lambda}}{\sqrt{2\hat{\Sigma}_y}} = \frac{3C_{\Lambda} \hat{\Lambda}}{\sqrt{2e^2h\tau_e\delta t}} \sqrt{\Pi_y\Pi_y} \tag{14}
\]

where \( \Pi_y = \sum_{\alpha} e_{\alpha}e_{\alpha}(f_{\alpha} - f_{\alpha}^{eq})^{\text{eq}} \).

Thus, the eddy relaxation time is

\[
\tau_e = \frac{3v_{e}}{\hat{\delta} t} = \frac{9C_{\Lambda} \hat{\Lambda}}{2e^2h(\tau_e + \tau)\delta t} \sqrt{\Pi_y\Pi_y} \tag{15}
\]

Here, if \( \hat{\Lambda} = \delta x \) is adopted, and then we can further write the above equation as

Now, we obtain the solution to the eddy relaxation by solving the above equation

$$\tau_i = \frac{9C_s}{\sqrt{2}e^h(r_i + \tau)} \sqrt{\Pi_x \Pi_y}$$  \hspace{1cm} (16)

Finally, the total relaxation time is

$$\tau_e = \tau + \tau_i = \frac{\sqrt{\tau^2 + \frac{36C_s}{\sqrt{2}e^h \sqrt{\Pi_x \Pi_y}}}}{2}$$  \hspace{1cm} (17)

The solution procedure for the lattice Boltzmann model of nonlinear shallow water governing equations is now summarized as follows:

1. Give initial water depth and velocity;
2. Calculate $\varphi_{eq}$ from Eq. (8);
3. Calculate the total relaxation time $\tau_e$ from Eq. (18);
4. Compute $f_e$ from Eq. (6);
5. Update the depth and the velocity according to Eq. (9);
6. Return to step (2) and iterate the above procedure until a solution is obtained.

2.3 Boundary and initial conditions

2.3.1 Solid wall boundary conditions

The basic idea of the bounce-back scheme is that an incoming particle towards the boundary bounces back into fluid. As an example of the upper solid wall shown in Fig. 2, the incoming known distribution functions $f_2$, $f_3$, and $f_4$ move towards the solid wall and they are immediately bounced back by the wall, and thus the unknown $f_6$, $f_7$, and $f_8$ after streaming can be simply determined by $f_6 = f_2$, $f_7 = f_3$, and $f_8 = f_4$. As a result, the sum of the particle moment close to the solid wall is zero, which indicates that this bounce-back scheme leads to the no-slip and no-penetration conditions.

2.3.2 Inlet boundary conditions

The uniform velocity and the uniform water depth are given at the inlet boundary. In this section a boundary condition is proposed based on the bounce-back scheme of the non-equilibrium part (Zou and He, 1997). The unknown distribution function $f_e$ at the inlet boundary is shown in Fig. 2. After streaming, $f_3$, $f_4$, $f_5$, $f_6$, $f_7$, and $f_8$ are known, and now, we want to use Eq. (9) to determine $f_1$, $f_2$, $f_9$, which can be put into the form

![Fig. 2 Boundary condition schemes](image-url)
To close the outlet boundary condition, we assume the bounce-back rule is still correct for the non-equilibrium part of the particle distribution normal to the inlet boundary in this case, i.e. \( f_1 - f_1^eq = f_5 - f_5^eq \). With \( f_5 \) known, \( f_1 \) can be found, and \( v \) is assumed zero, thus we obtain

\[
f_1 = f_5 + \frac{2hu}{3e} \]
\[
f_2 = \frac{hu}{6e} + f_5 + \left( \frac{f_5 - f_2}{2} \right) \]
\[
f_5 = \frac{hu}{6e} + f_5 + \left( \frac{f_5 - f_6}{2} \right) \]

2.3.3 Outlet boundary conditions

The gradient of the distribution function normal to the outlet boundary is set zero. After streaming, the unknown distributions \( f_4, f_5, \) and \( f_6 \) at the outflow boundary shown in Fig. 2 are simply calculated by

\[
f_a(N_x, j) = f_a(N_x - 1, j), (\alpha = 4, 5, 6, j = 1 - N_y) \]

where \( N_x \) and \( N_y \) are the total lattice number in the \( x \) direction and the \( y \) direction, respectively.

2.3.4 Initial conditions

Generally speaking, it is often easier to specify a macroscopic quantity than a microscopic one. So, the flow field is defined first, and then is used to calculate the local equilibrium distribution function \( f_0^eq \), which is used as an initial condition for \( f_0 \), i.e. \( f_0 = f_0^eq \). And the time step \( \delta t = 0.005s \).

3 Model setup

The experiment is conducted in Tsinghua University in a straight rectangular glass flume, 30-m long, 2-m wide and 0.12-m deep with a smooth bed and sides. A smooth rectangular glass groyne is fixed perpendicular to and along the flume side at a distance \( x = 15 \) m from the inlet. The groyne is 0.1-m wide in the longitudinal direction, 0.5-m long in the lateral direction and high enough to project above the water free surface as shown in Fig. 3. The measured region in the longitudinal direction includes a groyne length of 0.1 m, an upper groyne length of 2.8 m and a lower length of 2.42 m.

![Fig. 3 Test configuration](image)

At the downstream end of the flume, a tailgate is installed to regulate the depth of flow. Water is pumped from an underlying sump into the flume head tank. At the upstream cross section far away from the groyne, the gross discharge rate in the flume could be obtained via the velocity distribution, which reaches a fully-developed steady and uniform state undisturbed by the groyne. When the water flow in the flume remains steady, the mean water depth in the flume was 0.1024 m and the flow rate is \( 6.61 \times 10^{-2} \) m\(^3\)/s. And then the longitudinal and lateral velocity distributions around the groyne are measured using two-dimensional Laser Doppler Velocimeter (LDV) and the error of the maximum measurement range is less than 0.1%. Each vertical line at various cross-sections consists of 4 measurement points and the International Journal of Sediment Research, Vol. 25, No. 3, 2010, pp. 271–282 - 277 -
depth-averaged velocity distributions are calculated in the vertical direction. The water depth is recorded using point gauge and its error is not higher than 0.5%.

Considering all the factors which would probably affect the calculated results, an appropriate computational domain was adopted here, which is 10 m in the longitudinal direction where the length of the upper groyne is 3.8 m long and the length of the lower groyne is 5.1 m long. Several computational trials are run with various resolutions in order to ensure that final grid is sufficiently refined. Finally the grid system consists of $I \times J = 1,001 \times 201$ cells where $I$ is in the longitudinal direction and $J$ is in the lateral direction.

4 Model results

4.1 Velocity distributions

The flow patterns around the groyne are illustrated in Figs. 4-6. It can be seen that the main flow separates at the head of the groyne and constricts till it reaches the narrowest downstream, and then it moves towards the sidewalls adjacent to the groyne. A large secondary flow, one small vortex downstream from the groyne and another small vortex in the corners in front of the groyne are predicted. There is a large dead zone behind the groyne where the velocity is very low. The downstream velocity distributions far away from the groyne are predicted to gradually recover.
The calculated velocity components $U$ and $V$ around the groyne in the directions $X$ and $Y$ are compared quantificationally with the experimental as shown in Fig. 7. The velocities evolve when flowing around the groyne. The maximum computed velocity is 0.79 m/s and the maximum experimental velocity is 0.66 m/s, and the difference is probably due to the errors from the testing and relevant data-processing techniques. The calculated velocity distributions are in fairly good agreement with the experimental data. There is a little difference in the $X = 4.03m$ cross-section, where the site of the calculated maximum velocity component $U$ is much closer to the groyne than the experimental data.

4.2 Streamline distributions

The computational streamline distributions around the groyne are shown in Fig. 8. Both the calculated and the experimental recirculating length is about 3.69 m. All various vortexes around the groyne are predicted successfully. It can be seen from Figs. 5-6 and 8 that there is a large recirculating zone behind the groyne where the velocity components are very small, which must lead to sedimentations for silt-laden two-phase flows, whereas there is probably no sedimentation where there is a large velocity distribution.
4.3 Water level distributions

The computational water depth distributions around the groyne are demonstrated in Fig. 9. The water level in the upstream of the groyne is higher than that in the downstream of the groyne because of the groyne blockage and the water depth reaches the highest value. The water level gradient is much steeper around the groyne than anywhere else. Additionally, the water level far away from the groyne recovers gradually downstream.

The calculated depth at various cross-sections around the groyne is compared quantificationally with the experimental data as shown in Fig. 10. The numerical results agree well with the experimental data. The relative difference between both values is smaller than 0.5% all over data field. The water depths in front of the groyne reach the highest values, i.e., experimental value with 10.80 cm and calculated value with 10.71 cm.
4.4 Vorticity distributions

The computational vorticity distributions are shown in Fig. 11. The signs of the numerical vorticity indicate that the vortex rotating directions agree with the rule in nature. The analysis of the vorticity field presents a particular interest for studying the sediment transport around the groyne. The criterion of starting motion of particle is generally estimated from a critical velocity constituting a threshold. We proposed, as an example, to estimate zones of sediment deposit and transportation according to the velocity magnitude and vorticity. And then potential depositing zones are estimated from the velocity and vorticity distributions shown in Figs. 5 and 11 obtained during the simulation. It can be seen from Fig. 11 that there is an apparent shear layer near the maximum vorticity line, i.e. the recirculation line, which states that some particles put in motion out of the secondary region behind the groyne, will move here to rest downstream and more specifically along the recirculation line. We may analyze this in an example by employing well-known criteria for putting sediment into motion or not.

Fig. 11 Computational vorticity distribution (Unit 1/s)

5 Conclusions

Firstly, a lattice Boltzmann model coupled with a SGS model is proposed and is successfully applied in the flows around a groyne. The calculated velocity and water level distributions are both in fairly good agreement with the experimental data, and the computed recirculating length compared favorably with the experimental length. Large secondary flow occurs downstream from and caused by the groyne. The detailed flow information is obtained using numerical techniques. Two small vortexes in the corners in front of and behind the groyne are predicted, and the results indicate that the LBM-SGS model could give the fine flow pattern around the groyne. There is a dead zone behind the groyne where the velocity is very low, which leads to sedimentation for silt-laden two-phase flows. The groyne has little effect on the flows far away from the groyne in the lateral direction.

The resulting velocity and vorticity distributions are in good agreement with whether some sediment transportation phenomena or well-known criterion of starting motion.

The good agreement between the experimental data and the simulation shows that the LBM-SGS model and the numerical algorithm are practicable and credible. Further experiments and numerical simulations will be carried out in the future to investigate three-dimensional silt-laden flows around the groyne in the future.

Also it is also noted that a little differences in the velocity and water depth distributions occurs between the calculated results and the experimental data, which probably results from the errors from the testing and relevant data-processing techniques.

Acknowledgements

The work reported here is supported by the National Natural Science Foundation of China (No.50779069, No. 50976124) and the Beijing Natural Science Foundation (3083022), the Open Fund of Key Laboratory of Yellow River Sediment Research of the Ministry of Water Resources (No.200903), the National Science and Technology Supporting Programs in the 11th Five-Year Plan (2006BAB06B02), the Chinese Universities Scientific Fund (No. 2009-1-90, No. 2009-2-12) and the Program for Changjiang Scholars and Innovative Research Team in University (No.PCSIRT0657).
References