Simulation of Shallow Water Flows in Confluence Channel Using Lattice Boltzmann Method

Qing-Yuan YANG, Wei-Zhen LU[*a)]

[*] *Department of Civil and Architectural Engineering, City University of Hong Kong, Hong Kong SAR*

Summary This study presents a preliminary numerical investigation on shallow water flow in confluence river channel using the LBM approach. The aims of the investigation are detecting the separation zone and estimating its impact on downstream.

The lattice Boltzmann method (LBM) is regarded as an efficient, flexible approach for simulating flows with complex geometries. The method calculates the microscopic movement in a much simplified way, and, at the macroscopic level, it represents an average description of a fluid. Its simple calculation, parallel process and easy implementation make it a very promising computation method in many areas. However, very few applications can be traced in shallow water flows. This study presents a preliminary numerical investigation on shallow water flow in confluence river channel using the LBM approach with the aims of detecting the separation zone and its impact on downstream.

Shallow water flows exist in many practical situations such as tidal flow, waves, open channel flow and dam-break flow. As shown in fig. 1(a), the key characters of flow in confluence channel are the separation zone near the junction point and the section circumfluence along the downstream channel. When two channel flows meet at the confluence, due to the velocity difference, there is usually a shear plane in between, which is one of the major sources of turbulence generation. At the beginning of the shear plane is the stagnation zone. Downstream of the branch channel, a separation zone is usually generated, which is another source of turbulence. In the separation zone, the velocity magnitude is small and the velocity gradient near the edge of separation zone is large. Fig. 1(b) shows the three-dimensional view of the confluence flow (Webber et al [1]). Due to the differences in centrifugal forces along the channel depth, the secondary circulation is generated. The separation zone and the secondary circulation have significant impact on sediment transport and pollutant dispersion in confluence region. Fig. 2 presents the simulation results and comparisons to the experiments.

By solving the LBGK equation (Eq. 1) implemented with local equilibrium distribution function introduced by Zhou[2] and subgrid-scale stress (SGS) model, this study comes up with a numerical investigation of the flow structure in channel confluence. A finite volume approach is adopted to discretize the LBE on a cell-centered triangular grid. The advection terms is discretized with the second order, i.e., Total Variation Diminishing (TVD) scheme (Dhiraj et al [3]). With detailed experimental data, the results also show a detailed comparison between the simulation results and the relevant experimental data.

Simulation results show that, the LBM is simple and accurate for solving the shallow water flows and the simulation results agree well with the experiment data.

$$f_\alpha\left(x+\overrightarrow{e_\alpha}\Delta t, t+\Delta t\right)-f_\alpha\left(x,t\right)=-\frac{1}{\tau}\left(f_\alpha-f_\alpha^{eq}\right)+\frac{\Delta t}{N_\alpha e^2}e_{\alpha i}F_i \tag{1}$$

$$h\left(\vec{x},t\right)=\sum_\alpha f_\alpha\left(\vec{x},t\right)$$

$$u_i\left(\vec{x},t\right)=\frac{1}{h\left(\vec{x},t\right)}\sum_\alpha e_{\alpha i} f_\alpha\left(\vec{x},t\right) \tag{2}$$

$$F_i=-gh\frac{\partial z}{\partial x_i}-\frac{\tau_{bi}}{\rho} \tag{3}$$

Where f_α is the distribution function along the direction $\overrightarrow{e_\alpha}$, h is the depth of water, u is the depth averaged velocity, F is the body force, z is water surface elevation and τ_b is bed shear stress.

[a)] Corresponding author. Email: bcwzlu@cityu.edu.hk.

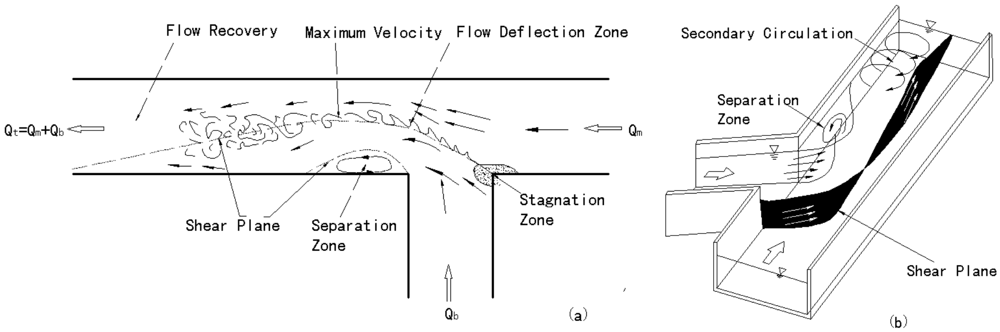

Figure 1: Flow Characteristics in Open-Channel Confluence

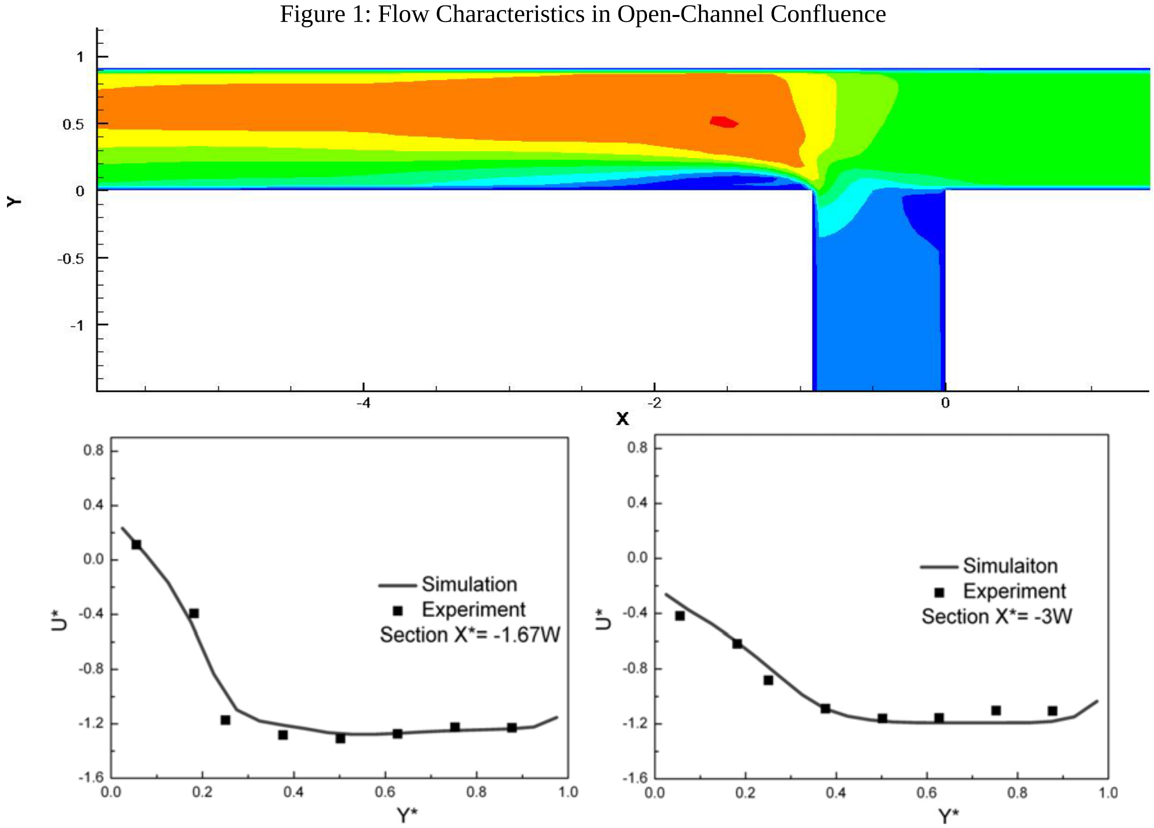

Figure 2 Mesh scheme and simulation results

CONCLUSIONS

The finite volume based LBM simulation can adopt the triangular mesh, polygonal mesh and mix of them. It is can be used to solve those problems with irregular boundary. The finite volume based LBM simulation inherits the advantage for parallel calculation and is good at simulations with a large number of meshes. The mesoscopic nature of LBM simulation, coupled with the subgrid stress model, make it easy to carry out Large Eddy Simulation and investigate the complicated flow structure in open channel flow.

Acknowledgments

The work was partially supported by the projects of Strategic Research Grant (Grant No. 7002684(BC)), City University of Hong Kong, HKSAR and the Open Foundation of State Key Laboratory of Hydraulics and Mountain River Engineering, Sichuan University (Grant No. 1008).

References

[1] Weber, L.J., Eric, D.S., and Nicola, M. Experiments on Flow at a 90° Open-Channel Junction. Journal of hydraulic engineering-ASCE 2001, 127(5):340-350

[2] .Zhou JG. Lattice Boltzmann methods for shallow water flows. Springer; Germany, 2004.

[3] Dhiraj V.P. and Lakshmisha K.N. Finite volume TVD formulation of lattice Boltzmann simulation on unstructured mesh. Journal of Computational Physics 2009, 228(14):5262-5279.