THE POTENTIAL OF THE LATTICE-BOLTZMANN METHOD IN COMPUTATIONAL HYDRAULICS

Zoltan Horvat [1]
Mirjana Horvat [2]
Budo Zindović [3]
Nikola Rosić [4]
UDK: 532:519.6
DOI:10.14415/konferencijaGFS 2016.073
Summary: *Although the lattice-Boltzmann method (LMB) originates from the kinetic theory of gases, it can be adopted for many practical engineering uses. This paper explores the potential of the LBM in computational hydraulics. The discrete lattice-Boltzmann equation is implemented using the appropriate equilibrium distribution function to simulate flow in a simple open channel. Numerical tests are conducted in order to assess the ability of the numerical model based on the LBM to reproduce steady state situations, slip and no-slip boundary conditions on impermeable walls, upstream and downstream boundary conditions commonly used in hydraulic engineering. The paper also presents the results of unsteady flow simulations, as well as flow simulations over a non-uniform channel bed.*

Keywords: *lattice-Boltzmann method, computational hydraulics, numerical tests*

1. INTRODUCTION

The Boltzmann equation is a kinetic equation that deals with the motion of fluids in meso-scale and it relies on statistical mechanics. Statistical mechanics predicts the way in which atoms and molecules (microscopic scale) determine the macroscopic properties of fluids, such as viscosity, pressure, etc. However, the Boltzmann equation replaces tagging each fluid particle with a distribution function, i.e. it describes the propagation of a distribution function instead of the propagation of each and every particle in a mass of fluid. This is the essence of describing fluid flow in meso-scale [1,2].
The lattice-Boltzmann method was developed from the lattice-gas automata [3]. The core principle of the lattice-Boltzmann method is the indirect solution of the fluid motion equations, by solving something else, something much simpler. Therefore, simple

[1] Zoltan Horvat, Civ.Eng., PhD, University of Novi Sad, Faculty of Civil Engineering Subotica, Kozaračka 2a, Subotica, Serbia, tel: +381 24 554 300, e-mail: horvatz@gf.uns.ac.rs
[2] Mirjana Horvat, Civ.Eng., PhD, University of Novi Sad, Faculty of Civil Engineering Subotica, Kozaračka 2a, Subotica, Serbia, tel: +381 24 554 300, e-mail: isicm@gf.uns.ac.rs
[3] Budo Zindović, Civ.Eng., MSc, University of Belgrade, Faculty of Civil Engineering, Bulevar Kralja Aleksandra 73, Belgrade, Serbia, tel: +381 11 3218 553, e-mail: bzindovic@hikom.grf.bg.ac.rs
[4] Nikola Rosić, Civ.Eng., MSc, University of Belgrade, Faculty of Civil Engineering, Bulevar Kralja Aleksandra 73, Belgrade, Serbia, tel: +381 11 3218 553, e-mail: nrosic@grf.bg.ac.rs

arithmetic calculations can generate accurate solutions to the complex partial differential equations that describe fluid flow. This paper presents a lattice-Boltzmann model for the shallow water equations commonly used in computational hydraulics.

2. GOVERNING EQUATIONS

The Boltzmann equation is a kinetic equation that describes the propagation of the distribution function and it is given as [3]

$$\frac{\partial f}{\partial t} + \vec{c} \cdot \nabla f + \frac{\vec{F}}{\rho} \cdot \nabla_c f = \Omega(f), \tag{1}$$

where f denotes the distribution function, t is the time parameter, $\vec{c}$ is the fluid particle velocity vector, ∇ marks the gradient in physical space, ∇_c is the gradient in velocity space, $\vec{F}$ is the external force vector, ρ marks the fluid density, while $\Omega(f)$ denotes the particle collision operator.

Unlike when solving partial differential equations using traditional numerical procedures, the Boltzmann equation has to be discretized both in physical and velocity space. Therefore, a lattice pattern is introduced, which has two functions: defining computational points (i.e. computational mesh size) and determining the fluid particles' velocities. In this paper a 9-speed square lattice is used (Fig. 1), where c_i, i = 0,1,...,8 denote discrete particle velocities.

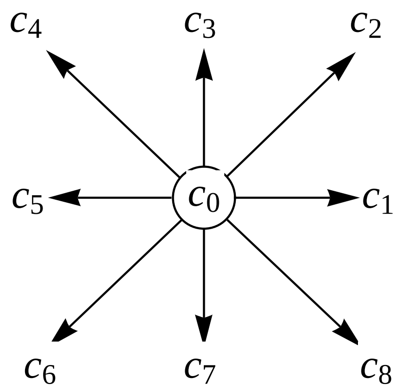


Figure 1. Lattice pattern

On this lattice pattern, each fluid particle moves one lattice unit at its velocity along the eight links (indexes 1 through 8 on Fig. 1) during each computational time step, while index 0 indicates the rest particle with zero speed. Therefore, the particle velocity vector can be defined as

$$\vec{c}_\alpha = \begin{cases} (0,0), & \alpha = 0 \\ c\left[\cos\dfrac{(\alpha-1)\pi}{4}, \sin\dfrac{(\alpha-1)\pi}{4}\right], & \alpha = 1,3,5,7 \\ \sqrt{2}\,c\left[\cos\dfrac{(\alpha-1)\pi}{4}, \sin\dfrac{(\alpha-1)\pi}{4}\right], & \alpha = 2,4,6,8 \end{cases} \tag{2}$$

where α is the lattice index consistent with notation in Fig. 1. Since the fluid particles move one lattice unit in each time step, the particle velocity intensity in Eqs. (2) is defined as

$$c = \frac{\Delta x}{\Delta t}, \tag{3}$$

where Δx and Δt respectively denote the computational space- and time-steps. Consequently, after discretizing the Boltzmann equation (1) both in physical and velocity space, and after implementing the Bhatangar, Gross and Krook (BGK) approximation on the particle collision operator, one can derive the lattice-Boltzmann equation

$$f_\alpha\left(\vec{x} + \vec{c}_\alpha\,\Delta t, t + \Delta t\right) - f_\alpha\left(\vec{x}, t\right) = -\frac{1}{\tau}\left(f_\alpha - f_\alpha^{(0)}\right) + \frac{\Delta t}{N c^2} c_{\alpha i} F_i\,, \tag{4}$$

where $f^{(0)}$ is the equilibrium distribution function, τ is the relative relaxation time, index i marks the summation index consistent with Einstein's notation, and N denotes the lattice number defined as

$$N = \frac{1}{c^2}\sum_\alpha c_{\alpha i}\, c_{\alpha i}\,. \tag{5}$$

The problem now boils down to determining the appropriate equilibrium distribution function in such a way that the lattice-Boltzmann equation (4) can recover the macroscopic shallow water equations. The theory of the lattice-gas automata clearly states that an appropriate function for this role is the Maxwell-Boltzmann equilibrium distribution function [3]. However, the use of this equilibrium function in Eq. (4) implies that the lattice-Boltzmann equation can recover only the Navier-Stokes equations. Thus, an alternative approach is to assume an equilibrium function as a power series of the macroscopic velocity

$$f_\alpha^{(0)} = A_\alpha + B_\alpha c_{\alpha i} u_i + C_\alpha c_{\alpha i} c_{\alpha j} u_i u_j + D_\alpha u_i u_i , \tag{6}$$

where u_i marks the macroscopic fluid velocity, while coefficients A, B, C and D remain to be determined. Since the equilibrium function has the same symmetry as the lattice pattern (Fig. 1), there must be

$$A_1 = A_3 = A_5 = A_7 = \bar{A}, \qquad A_2 = A_4 = A_6 = A_8 = \tilde{A}, , \tag{7}$$

and similar expressions for coefficients B, C and D. The aforementioned coefficients in Eq. (6) can be determined using the constraints imposed on the equilibrium distribution function (mass and momentum conservation). For the shallow water equations, these constraints are given as

$$\begin{gathered}
\sum_\alpha f_\alpha^{(0)}(\vec{x},t) = h(\vec{x},t), \\
\sum_\alpha c_{\alpha i} f_\alpha^{(0)}(\vec{x},t) = h(\vec{x},t) u_i(\vec{x},t), \\
\sum_\alpha c_{\alpha i} c_{\alpha j} f_\alpha^{(0)}(\vec{x},t) = \frac{1}{2} g h^2(\vec{x},t)\delta_{ij} + h(\vec{x},t) u_i(\vec{x},t) u_j(\vec{x},t) ,
\end{gathered} \tag{8}$$

where h is the water depth, g is the gravitational acceleration, and δ_{ij} denotes the Kronecker symbol. Substituting Eq. (6) into Eqs. (8), while using expressions similar to Eq. (7), finally yields the equilibrium distribution function for Eq. (4) that enables the recovery of the shallow water equations.

$$f_\alpha^{(0)} = \begin{cases} h - \dfrac{5 g h^2}{6 c^2} - \dfrac{2h}{3c^2} u_i u_i , & \alpha = 0 \\[2ex] \dfrac{g h^2}{6 c^2} + \dfrac{h}{3c^2} c_{\alpha i} u_i + \dfrac{h}{2c^4} c_{\alpha i} c_{\alpha j} u_i u_j - \dfrac{h}{6c^2} u_i u_i , & \alpha = 1,3,5,7 \\[2ex] \dfrac{g h^2}{24 c^2} + \dfrac{h}{12 c^2} c_{\alpha i} u_i + \dfrac{h}{8 c^4} c_{\alpha i} c_{\alpha j} u_i u_j - \dfrac{h}{24 c^2} u_i u_i , & \alpha = 2,4,6,8 \end{cases} \tag{9}$$

In order to prove that the water depth and velocities computed from Eqs. (8), using the values of the distribution function yielded by Eq. (4), are indeed the solution of the shallow water equations, the Chapmann-Enskog expansion is performed. This procedure utilizes taking the Taylor expansion in time and physical space of the first term on the left hand side of Eq. (4), as well as the expansion of the distribution function around its equilibrium according to perturbation theory. After this, by grouping the terms of the same order of magnitude, one can recover the continuity equation for shallow water flow

$$\frac{\partial h}{\partial t}+\frac{\partial}{\partial x_j}\left(h u_j\right)=0\,, \tag{10}$$

as well as the momentum equation for shallow water flow

$$\frac{\partial}{\partial t}\left(h u_i\right)+\frac{\partial}{\partial x_j}\left(h u_i u_j\right)=-g\,\frac{\partial}{\partial x_i}\left(\frac{h^2}{2}\right)+\nu\,\frac{\partial^2}{\partial x_j \partial x_j}\left(h u_i\right)+F_i\,, \tag{11}$$

where the kinetic viscosity is defined as

$$\nu = c^2\,\Delta t\left(2\tau-1\right)/6\,, \tag{12}$$

while the force term is

$$F_i=-g\,h\frac{\partial z_b}{\partial x_i}-\frac{\tau_{bi}}{\rho}\,. \tag{13}$$

In Eq. (13) z_b denotes the bed elevation, and τ_b is the friction on the bed surface. It should be noted that Eq. (12) demonstrates a peculiar property of the lattice-Boltzmann method. Namely, the viscosity is dependent on the time step, as well as the mesh size (through the lattice velocity intensity c). Therefore, while performing numerical simulations, the relative relaxation time τ should be adjusted so that Eq. (12) gives the desired viscosity for a given mesh size and computational time step.

3. NUMERICAL RESULTS

Using the governing equations given in Section 2, a computer code for simulation of shallow water flow in a simple straight channel was developed using the lattice-Boltzmann method. Some numerical tests, performed in order to asses the developed code, are presented in this section.

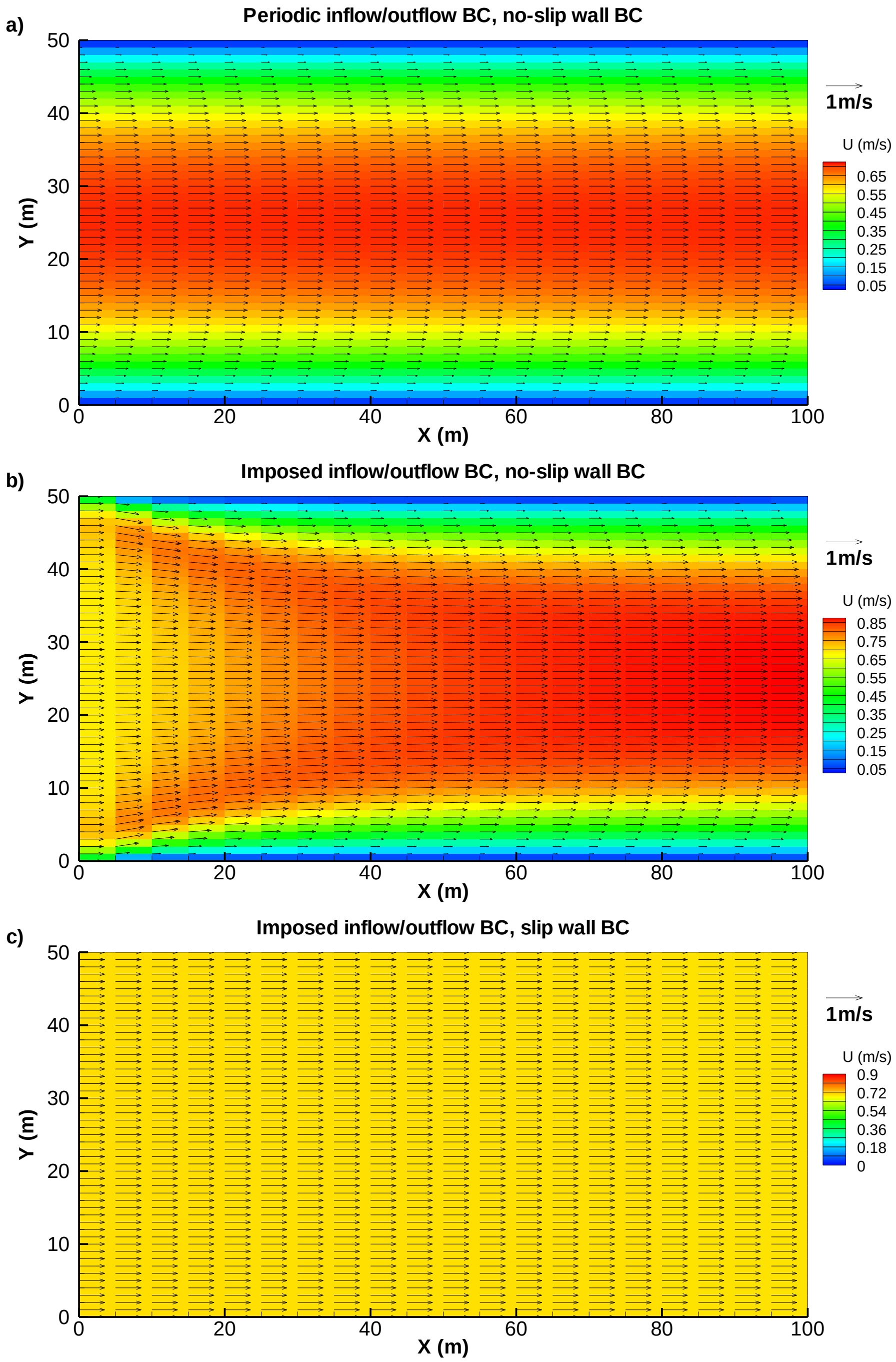


Figure 2. 2-D plots of steady flow in a channel with uniform bed elevation

The first group of numerical tests is based on steady flow simulations in a straight channel with uniform bed elevation. The computational domain was 100m in the *x* coordinate direction and 50m in the *y* coordinate direction. The computational mesh is uniform with Δx=1m, while the computational time step was selected in accordance with the stability conditions given in Ref. [4]. The computer code was developed in a way that enables the implementation of slip and no-slip boundary conditions on impermeable walls using the bounce-back scheme as described in Ref. [4,5].

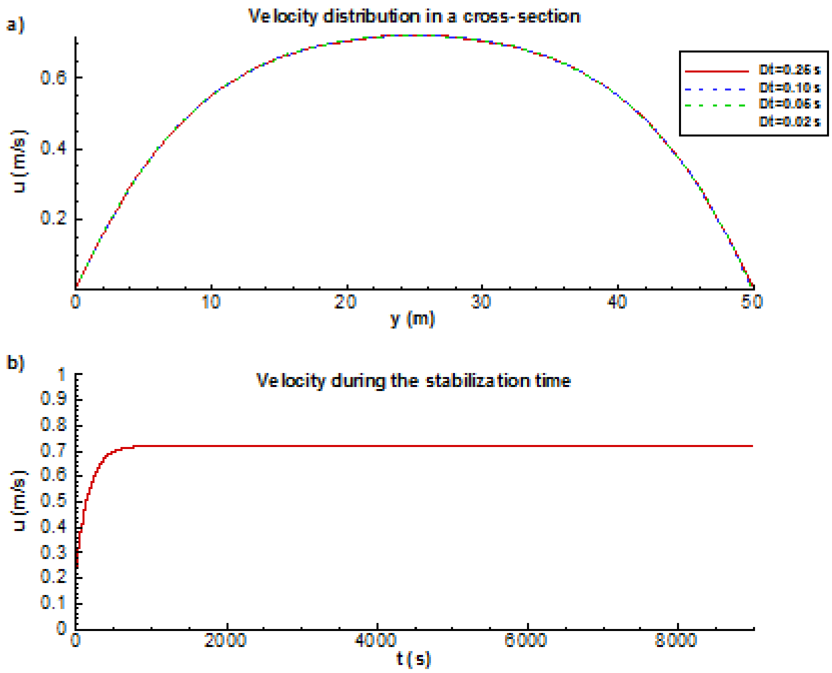


Figure 3. Velocity distribution and stabilization time

The firs simulation in this group utilizes the periodic boundary condition on the upstream and downstream ends, along with the no-slip boundary condition on the domain walls. A 2-D plot of this computation is presented on Fig. 2a. In this case the flow pattern consists of zero velocity on the impermeable walls (no-slip b.c.) and maximum velocities at the midpoint of any cross-section (straight line perpendicular to the *x*-coordinate direction, i.e. *x*=*const.*). The ability of the model to reproduce the same velocity distribution for different computational time step was also investigated. Figure 3a demonstrates that there is no significant difference in numerical results even if the computational time step is decreased more than 120%. Since the purpose of this simulation was to achieve a steady state flow, the evolution of fluid velocity through time was also investigated (Fig. 3b). It should be noted that, after achieving a steady state, the continuity error was reduced to practically zero (i.e. machine precision). The aforementioned properties indicate a certain consistency of the model.

In practical hydraulic computations the upstream boundary condition is often a known discharge, while a known water surface elevation is imposed on the downstream end. The developed computer code also enables these boundary conditions on the upstream and downstream ends of the computational domain, respectively. Therefore, the first simulation was repeated with known discharge at the upstream end and known depth on the downstream end, while implementing the no-slip boundary condition on impermeable walls. A 2-D plot of this computation is presented on Fig. 2b. It is clear that the model redistributes the imposed discharge, which also indicates a good qualitative behavior of the model. Finally, a steady state computation was performed with a slip boundary condition on impermeable walls and imposed discharge and water surface level on upstream and downstream boundaries. A 2-D plot of this computation is presented on Fig. 2c. It can be clearly observed that the simulation resulted in a uniform velocity distribution throughout the domain, as expected.

The second group of numerical test aimed to assess the ability of the model to perform unsteady flow computations. Using the domain and boundary conditions as on Fig. 2b, an unsteady flow simulation was conducted. A synthetic hydrograph was implemented at the upstream end, while a constant water surface elevation was maintained at the downstream end. The velocity change through time for a computational point at x=50m, y=25m is presented on Fig. 4. It can be concluded that after the stabilization time (1800 sec) the hydrograph is propagated through the domain with negligible numerical oscillations. The continuity error was monitored during the simulation, and it stayed under $4 \cdot 10^{-5}$%. Figures 4b and 4c present, in some detail, the numerical oscillations during the stabilization time and when the hydrograph reached its maximum value.

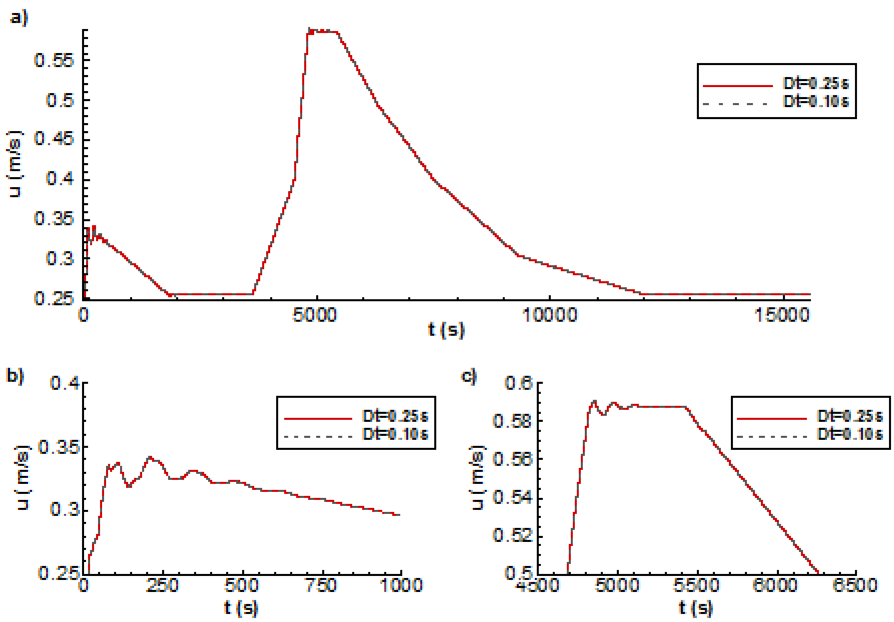
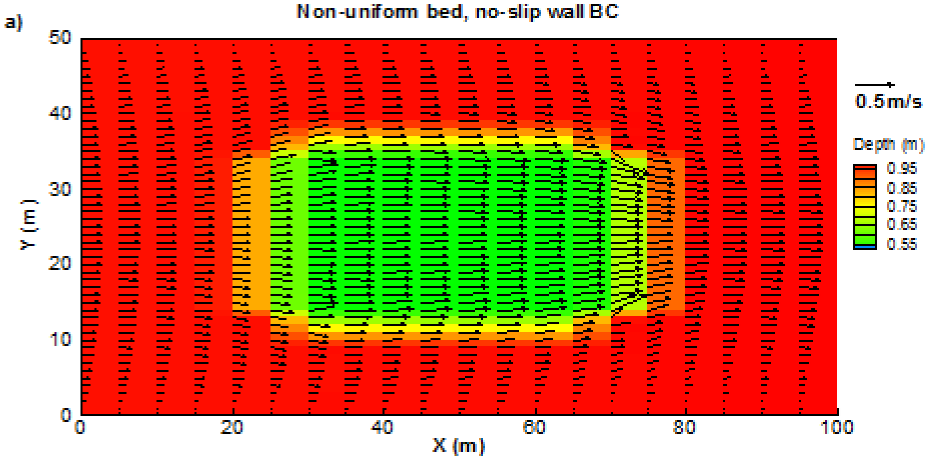


Figure 4. Unsteady flow computation

The third, and final group of numerical tests consisted of flow simulations in a channel with non-uniform bed elevation. In the interior of the computational domain the bed elevation is somewhat higher in comparison with the surrounding area, thus creating an obstacle to the flow. A 2-D plot of this computation is presented on Fig. 5a. Since a no-slip boundary condition was implemented on domain walls, the velocity is zero on the wall itself and it increases towards the interior of the domain. The velocity is significantly higher over the elevated bed area, as expected. Figure 6a presents the velocity distribution within a cross-section at the beginning (x=24m), at the middle (x=50m) and at the end (x=70m) of the obstacle. The existence of a v-velocity component indicates the tendency of the flow to bypass the elevated bed area, since it presents a restriction to the flow (higher bed friction, etc.).



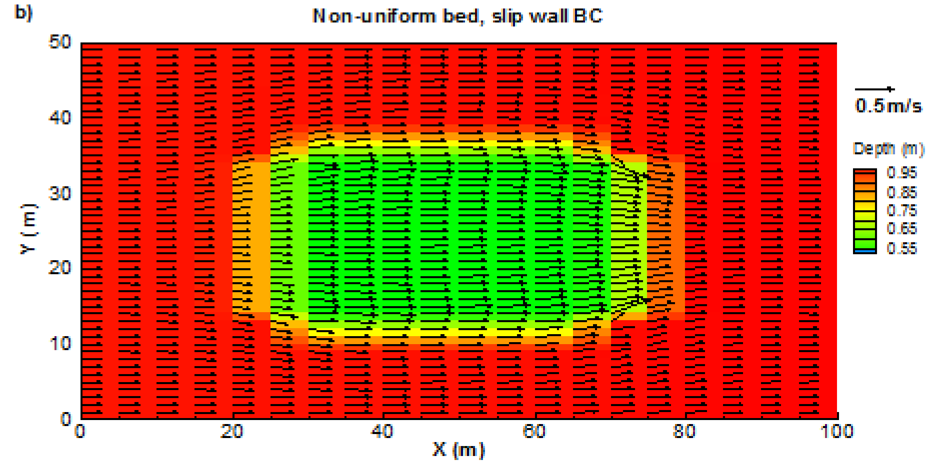


Figure 5. 2-D plots of steady flow in a channel with non-uniform bed elevation

The second simulation in this group was identical to the first, except for the impermeable wall boundary condition which was set to a slip boundary condition. A 2-D plot of this computation is presented on Fig. 5b, while the appropriate cross-section velocity distributions are given on Fig. 6b. The results are similar as in the previous case, with the exception that the flow is now more explicitly redirected toward the domain walls. This is also an expected behavior of the model, since the velocity on the impermeable boundary is not set to be zero. Therefore, the fluid is not additionally slowed down in the vicinity of the domain walls.

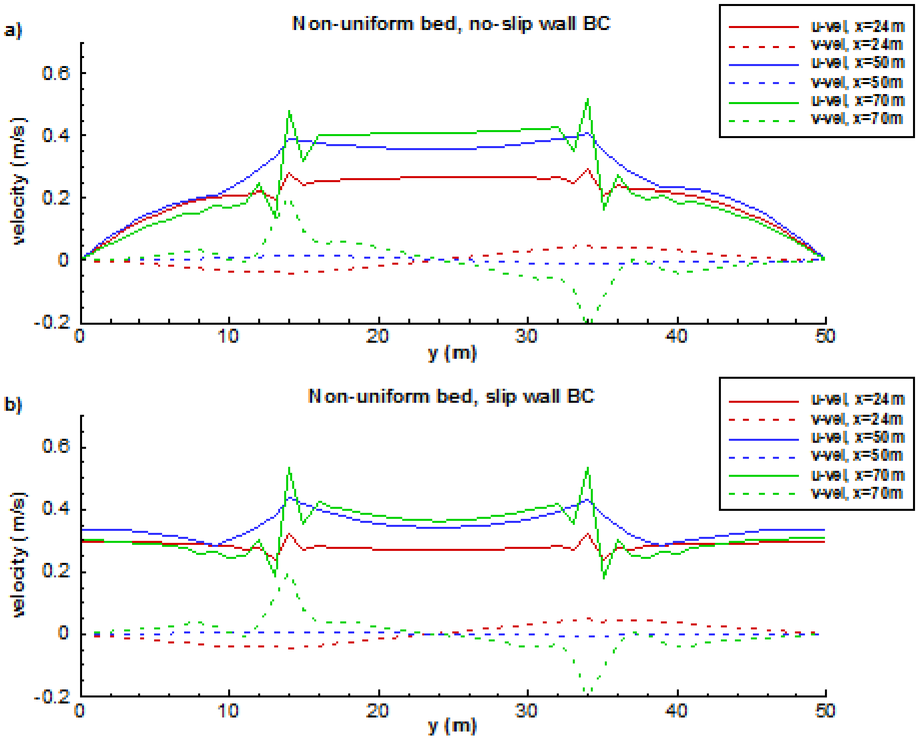


Figure 6. Velocity distribution for flow simulation in a channel with non-uniform bed elevation

4. CONCLUSION

Unlike the traditional numerical models in computational hydraulics that directly solve the shallow water equations, the lattice-Boltzmann method utilizes an indirect way to solve these equations. The method's trademark is a simple calculation procedure and easy implementation of boundary conditions.

This paper describes a numerical model for shallow water flow simulations using the lattice-Boltzmann method. The presented numerical tests were conducted in order to assess the basic properties of the numerical model as its' ability to achieve steady state flow conditions, unsteady flow simulations as well as the capacity to simulate flow over a non-uniform bed. It has been shown that the lattice-Boltzmann method is a very promising computational method that could be implemented in various aspects of hydraulic engineering. The main conveniences of the presented method are the following: only simple arithmetic calculations are used, the model uses only a single scalar variable, and finally, the governing equations are explicit which is ideal for parallel programming.

Although the developed code is capable of simulating only relatively simple flow fields, the authors believe that further research on this subject would be most advantageous.

REFERENCES

[1] Mohamad, A. A.: *Lattice-Boltzmann Method – Fundamentals and Engineering Applications with Computer Codes*, Springer, London, **2011**.
[2] Satoh, A.: *Introduction to practice of molecular simulation*, Elsevier, London, **2011**.
[3] Wolf-Gladrow, D. A.: *Lattice-Gas Cellular Automata and Lattice Boltzmann Models – An Introduction*, Springer, Berlin, **2005**.
[4] Zhou, J. G.: *Lattice Boltzmann Methods for Shallow Water Flows*, Springer, Berlin, **2004**.
[5] Zou, Q., He, X.: On pressure and velocity boundary conditions for the lattice Boltzmann BGK model. *Physics of Fluids*, **1997.**, vol. 9, № 6, p.p. 1591-1598.

ПОТЕНЦИЈАЛ LATTICE-BOLTZMANN МЕТОДЕ У НУМЕРИЧКОЈ ХИДРАУЛИЦИ

Резиме: *Иако lattice-Boltzmann-ова метода (ЛБМ) потиче из кинетичке теорије гасова, она се може применити и за практичне инжењерске прорачуне. Овај рад истражује потенцијал примене ЛБМ у рачунској хидраулици. Дискретна lattice-Boltzmann-ова једначина се примењује на течењу у једноставном отвореном каналу користећи одговарајућу равнотежну функцију расподеле. Спроведени су нумерички тестови у циљу стицања увида у могућности нумеричког модела да репродукује устаљено течење, граничне услове на непропусним зидовима са клизањем и без клизања, као и најчешће коришћене узводне и низводне граничне услове у хидротехници. Рад такође приказује резултате симулација неустаљеног струјања, као и резултате симулација течења преко неуниформног корита.*

Кључне речи: *lattice-Boltzmann метода, нумеричка хидраулика, нумерички тестови*