COMPUTATIONAL MECHANICS 2021
36th conference with international participation
Srní
November 8 - 10, 2021

Numerical simulation of flows in discharge objects with siphon

J. Fürst[a], T. Halada[a], M. Sedlář[b], T. Krátký[b], P. Procházka[c], M. Komárek[b]

[a] *Faculty of Mechanical Engineering, Czech Technical University in Prague, Karlovo nám. 13, 121 35 Praha, Czech Republic*
[b] *Centre of Hydraulic Research, Jana Sigmunda 313, 783 49 Lutín, Czech Republic*
[c] *Institute of Thermomechanics, Czech Academy of Sciences, Dolejškova 1402/5, 182 00 Praha, Czech Republic*

The paper deals with the numerical simulations of free-surface flows in discharge objects with siphon. The goal of the work is to investigate the applicability of the lattice Boltzmann method (LBM) for numerical simulations of the problem. The results of the LBM simulations are compared to numerical results obtained with the standard finite volume method in the Volume-of-Fluid formulation and with the experimental data obtained in the hydraulic laboratory with water circuit [4].

We consider a free-surface flows in a test channel depicted in Fig. 1. The water enters through the siphon at the left hand side of the channel and leaves the channel through an adjustable gate at the right hand side. The numerical solution is calculated in a computational domain corresponding to left part of the channel.

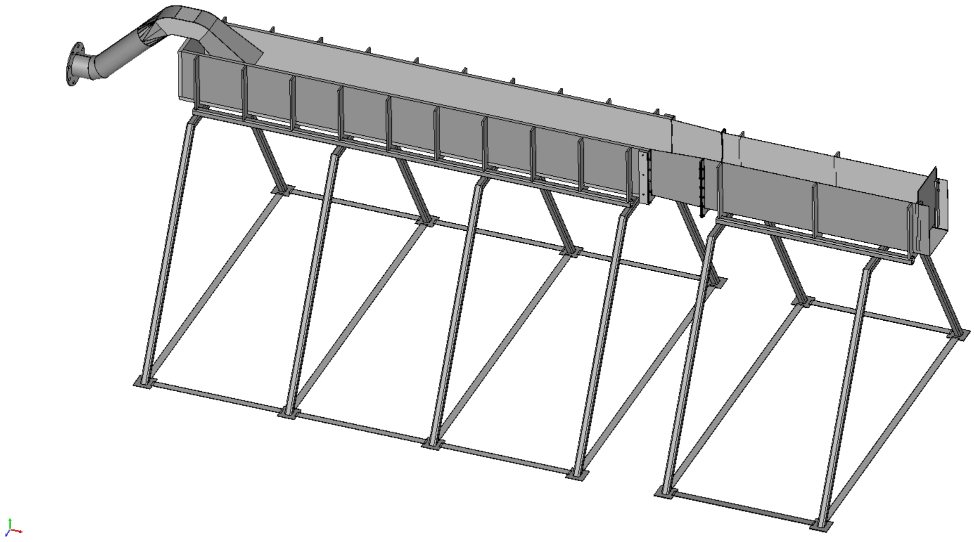

Fig. 1. Test channel with siphon

The numerical solution is obtained with the free surface LBM method proposed in [2] and carried out with an in-house code based on the open source LBM framework Palabos [3]. The motion of the liquid is described by the system of advection-reaction PDEs for the so-called discrete density distribution functions $f_i(\vec{x}, t)$

$$\frac{\partial f_i}{\partial t} + \vec{c}_i \cdot \nabla f_i = \frac{1}{\tau_c}\left(f_i - f_i^{(eq)}\right) + F_i, \tag{1}$$

where $\vec{c}_i$ are the discrete velocities of the D3Q19 model (see [2]), τ_c is the relaxation time, F_i represent the contribution of the external forces and $f_i^{(eq)}$ are the equilibrium distribution functions for incompressible fluid

$$f_i^{(eq)} = w_i \left[\rho + \frac{\vec{c}_i \cdot \vec{u}}{3} + \frac{(\vec{c}_i \cdot \vec{u})^2}{18} - \frac{\vec{u} \cdot \vec{u}}{6} \right], \tag{2}$$

where w_i are the weights of the D3Q19 model. The free-surface extension is described in [2]. The relaxation time τ_c is a function of the effective viscosity including the molecular and eddy viscosity calculated with the Smagorinsky model, for more details see [1].

The problem was solved using a Cartesian mesh with uniform spacing $\Delta x = \Delta y = \Delta z = 2\,\mathrm{mm}$ with staircase approximation of boundary and with constant time step $\Delta t = 5 \times 10^{-5}\,\mathrm{s}$. Although the staircase approximation with standard bounce-back boundary conditions impairs the overall accuracy of the method, the current study focuses mainly on the flow structures in the channel with straight boundaries. The parabolic velocity profile with given volumetric flow rate is prescribed ad the inlet and simple extrapolation is used at the outflow. The desired water level in the channel is kept using an adjustable gate near the outlet. The model does not use any local mesh refinement near the wall neither any wall functions. This simplification partially impairs the accuracy of the model near the wall and the fails to predict correct wall shear stresses. Nevertheless, the dynamics structure of the flow field further from wall corresponds very well to the numerical results obtained with the finite volume approach using the ANSYS CFX software and to the experimental data acquired with the laser particle image velocimetry (PIV) method.

Fig. 2 shows the outline of the domain and the water level calculated with LBM for prescribed flux rate $q = 13.8\,\mathrm{l/s}$.

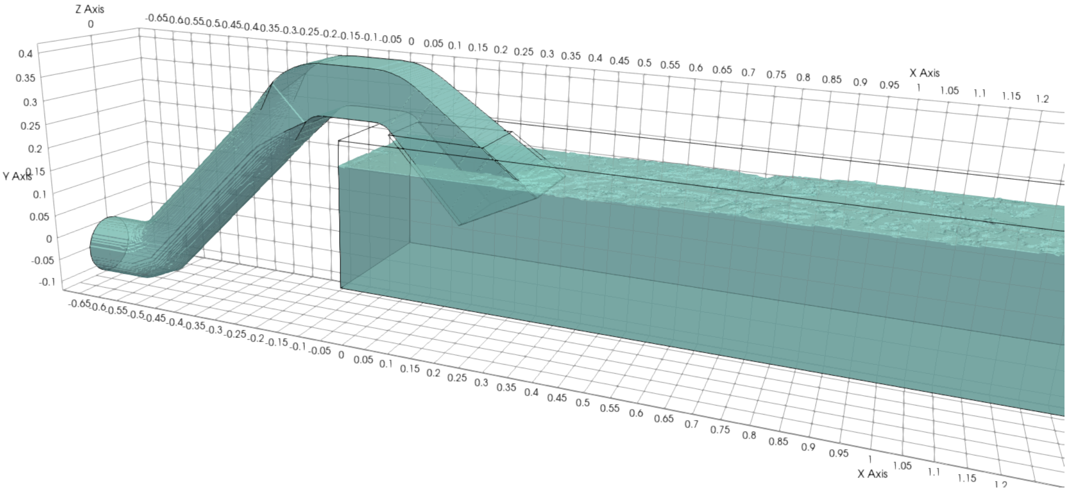

Fig. 2. LBM, water level at time $t = 30\,\mathrm{s}$ for flux rate $q = 13.8\,\mathrm{l/s}$

Fig. 3 shows the instantaneous magnitude of 2D velocity projection in vertical symmetry plane and the structure of the flow field using the line integral convolution (LIC) method. Since the method corresponds to LES approach, the calculated instantaneous flow field contains a lot of turbulent vortices. Therefore, the results were post-processed using standard time averaging over a time window $T = 15\,\mathrm{s}$ to $20\,\mathrm{s}$ and the time averaged results are plotted in Fig. 4.

Fig. 5 shows the results of PIV measurements. One can see very similar flow features, namely the vortex near the bottom at $x \approx 350\,\mathrm{mm}$ and the attachment point at $x \approx 500\,\mathrm{mm}$.

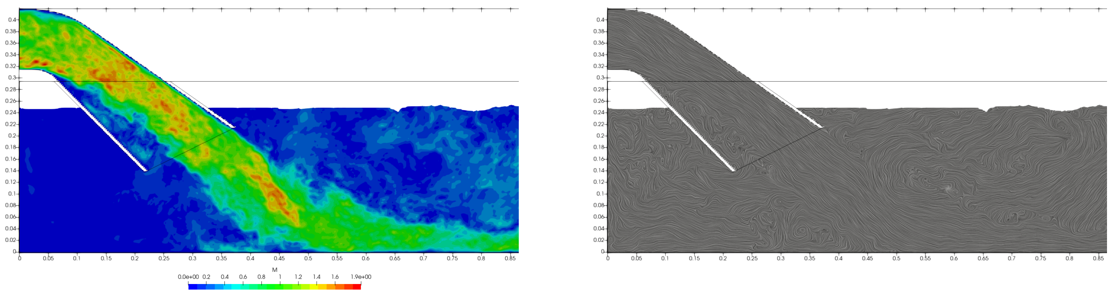

Fig. 3. Instantaneous 2D velocity magnitude (*left*) and flow field structure (*right*) in the symmetry plane

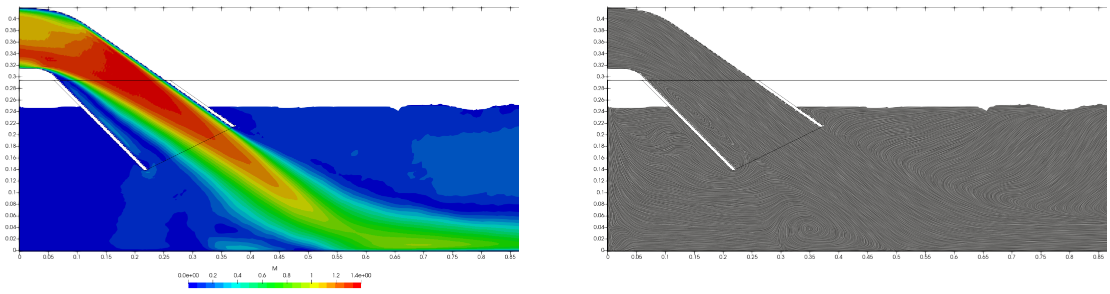

Fig. 4. Time averaged 2D velocity magnitude (*left*) and flow field structure (*right*) in the symmetry plane

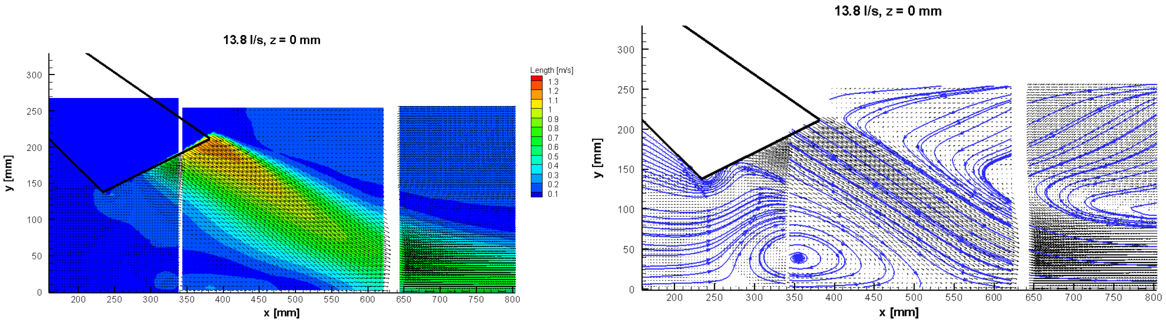

Fig. 5. PIV, 2D velocity magnitude and streamlines in the vertical symmetry plane $z = 0\,\mathrm{mm}$

Figs. 6 and 7 compare the calculated time averaged flow field with the PIV data in the horizontal plane $10\,\mathrm{mm}$ above the channel bottom. One can clearly see that the flow is non-symmetric. See, e.g., the position of the critical point ($x \approx 480\,\mathrm{mm}$). Note that the flow is highly unsteady and the calculated and measured positions of the critical point depend (among others) on the time averaging interval.

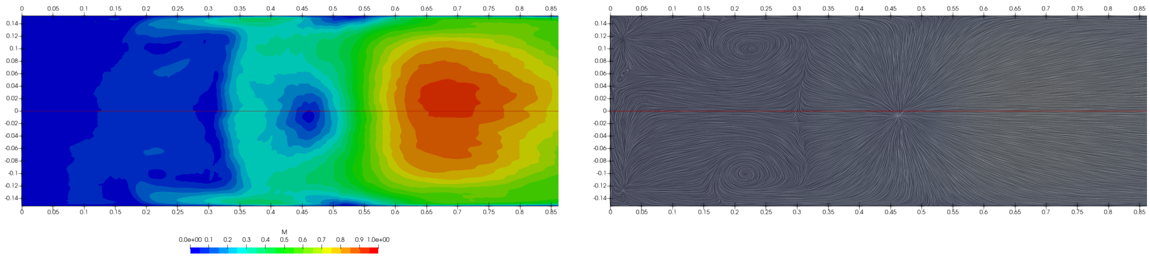

Fig. 6. Time averaged 2D velocity magnitude (*left*) and flow field structure (*right*) in the horizontal plane $10\,\mathrm{mm}$ above the bottom

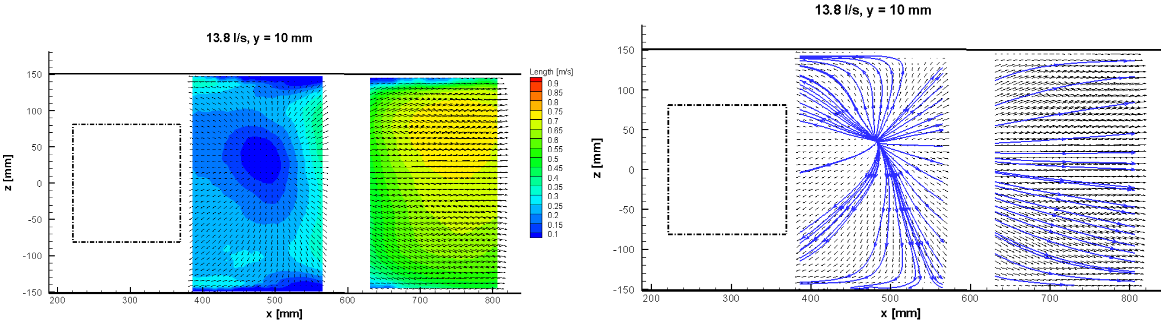

Fig. 7. PIV, 2D velocity magnitude and streamlines in the horizontal plane $y = 10\,\mathrm{mm}$

The paper shows that the free-surface LBM method provides a viable alternative to standard finite volume method (FVM) for given problem. The advantage of LBM over the FVM is its simplicity especially in the pre-processing phase. Moreover, the LBM is extremely easy to parallelize and can exploit modern CPU/GPU architectures. On the other hand, the implementation of boundary conditions in the case of non-planar boundaries becomes more difficult. Moreover, the current software package does not use any wall model (e.g., wall functions) and therefore it does not handle the very near wall flows correctly.

Acknowledgements

The work was supported by the Grant Agency of the Czech Technical University in Prague, grant No. SGS19/154/OHK2/3T/12 and by the Grant Agency of the Czech Technical University in Prague, grant No. SGS19/154/OHK2/3T/12.

References

[1] Fürst, J., Halada, T., Sedlář, M., Krátký, T., Procházka, P., Komárek, M., Numerical analysis of flow phenomena in discharge object with siphon using lattice Boltzmann method and CFD, Mathematics 9 (15) (2021) No. 1734.

[2] Körner, C., Thies, M., Hofmann, T., Thürey, N., Rüde, U., Lattice Boltzmann model for free surface flow for modeling foaming, Journal of Statistical Physics 121 (2005) 179-196.

[3] Latt, J., Malaspinas, O., Kontaxakis, D., Parmigiani, A., Lagrava, D., Brogi, F., Ben Belgacem, M., Thorimbert, Y., Leclaire, S., Li, S., Marson, F., Lemus, J., Kotsalos, C., Conradin, R., Coreixas, C., Petkantchin, R., Raynaud, F., Beny, J., Chopard, B., Palabos: Parallel lattice Boltzmann solver, Computers & Mathematics with Applications 81 (2021) 334-350.

[4] Sedlář, M., Procházka, P., Komárek, M., Uruba, V., Skála, V., Experimental research and numerical analysis of flow phenomena in discharge object with siphon, Water 12 (12) (2020) No. 3330.