UDC 519.63;.519.684

*Issakhov A.A., Mussakulova G.M.

Faculty of Mechanics and Mathematics,
al-Farabi Kazakh National University, Almaty, Kazakhstan
*e-mail: alibek.issakhov@gmail.com

Numerical simulation of the water flow in the case of an emergency at the protective hydraulic engineering constructions

Abstract. This paper presents numerical simulations of the water flow in the case of an emergency at the protective hydraulic engineering constructions. A mathematical modelfor simulation of water flow is based on Navier-Stokes equations derived from the conservation mass law and with phase equation. Numerical discretization of system of equations is done by finite volume method. System of water flow is solved by projection method. The phase and momentum equations are solved by Runge-Kutta method, and for the solution of Poisson equation the Jacobi method is applied. A program code was developed in C++ for numerical realization of water flow at emergency at the protective hydraulic engineering constructions.Numerical results are illustrated by figures.
Key words: Navier-Stokes equations, phase equation, mathematical model, hydraulic constructions, dam break.

Introduction

Dams are the structures designed to partition a watercourse or water for the lifting of the water level, in order to concentrate a water pressure at the site of construction and to createa reservoir. Typically, damsareparts of the hydroelectric complex, i.e., hydraulic structures, which are built in a particular location for the use of water resources for specific purposes: irrigation, hydropower, irrigation of pastures and so on.

The type and design of the dam are determined by its size, purpose, and the natural conditions and the main building material. Dams differ in type of the basic material from which they are erected, purpose and water passing conditions.

By the purpose of use dams are divided into storage reservoir and water level pumping types. Water pumping dams are built to improve the conditions of water intake from the river, the use of water power and so on. Therefore,affluent of the waterlevel of this type of dams is low. Storage dams differ markedly greater height, as a result, a large volume of the created reservoir. A distinctive feature of large storage dams is the ability to regulate the flow, small dams, by which, for example, ponds are created, don’t regulate the flow.

As an alternative to division of dams by theirs purposes acts division of dams by the height of water rise: low-pressure (depth of the water before the dam to 15 m), middle-pressure (15-50 m), high-pressure (over 50 m).

By the type of the material dams are divided into ground, tabby, metal, fabric, wood, iron-tabby, gabion types. By the way of perception of the main loads: gravity, arch, buttress, arch-gravity. By the conditions of the water flow passing: deaf, which do not allow overflow of the water through the crest spillway, filter, overflow.

Nowadays, potential catastrophic floods, resulting as an outcome of dam destructions, make great concern, since they bring big damage. The proof of this is seen in already occurred accidents. In 2010, a dam burst due to heavy rains took place on the Fuhe River in Jiangxi province in eastern China. One way or another, the disaster affected 29 million people. In the same year, there was a dam break on the Indus River in southern Pakistan. Over 895 thousand houses were destroyed, more than 2 million hectares of agricultural land were flooded. More than 1,700 people died. A year later, there was also a dam break, but this time, it took place on the Qiantang River near the city of Hangzhou in Zhejiang province in eastern China.Tidal wave of height up to 9 meters burst the dam and washed away a lot of people. Catastrophe at the dams, unfortunately, also took place in Kazakhstan too. On the 11th of March2010 due to a dam break in Kyzylagash several villages were flooded. 43 people died, 300 people were injured varying degrees of severity, and about 1,000 people were evacuated. Later the same year, there was another dam burst, but this time on the Usek River. 2187 inhabitants were evacuated. After 4 years a dam break took place in the village Kokpekty in the Karaganda region. As a result, five people died, 300 houses were flooded, 125 people were rescued, 300 were evacuated. And in 2015, as a result of a breakthrough the moraine lake Nameless, 40000 m^3 water was spewed. The mudflow came to the dam on the river Kargalinka. Three districts of the city were partially flooded, the Nauryzbaysky region was badly damaged.

As can be seen, dam accidents bring great damage, lead to the loss of human and financial resources. In order to minimize possibilities of such accidents a mathematical modeling is used, which allows to make small financial cost experiments, which results are very close to reality.

With the help of mathematical modeling, as well as the information about the area in which an artificial dam is planned to be built, an appropriate type of dam can be selected, the desired specifications of the dam, such as length, height and width of the walls, and so on, also can be selected. Mathematical modeling is also used in cases of accidents at the dams, to maximize the "softening" of aftermath. Selection of the most appropriate model and methods for simulating floods, caused by breakout, are very important steps. Also one should take into account to account that one-dimensional and two-dimensional models, compared tothree-dimensional model, have limitations, such as the failure of the first to embrace the spatial extent of the flood, from the standpoint of flow depth, velocity, time of arrival and flood recession, etc. These shortcomings are well studied in [1], which compares two-dimensional (Shallow water approach) and a full three-dimensional model. The latest model is composed by RANS equations, coupled with the method of volume of fluid. Comparisons of experimental and numerical data are also provided in the paper [1]. The results also show the superiority of the full three-dimensional model compared to the Shallow water approach.

Nowadays, as a result of development of technologies, parallel computation is also developed, which, as it is known, allows to spend less time for calculations. Thus, in [2], a mathematical model of a dam break problem is also provided and developed an improved method for correction of the pressure, in conjunction with the methods of volume of the liquid and the immersed boundary, to improve calculations of multiphase flow. TFQMR method is used to reduce the processing time in the solution of the Poisson equation.

In addition to the methods described in the papers mentioned above, there are also many other numerical methods for the dam break problems. So the method of integral boundaries are described in [3-5], method front track [6-8], the method of volume fluid [1, 2, 9, 10],the lattice Boltzmann method [11-13], the method the specified level [14-16], the phase-field methods [17-19].

Statement of the problem

The Navier-Stokes equations, derived from the law of conservation of mass, as well as the phase equation compose a mathematical model for the numerical simulation of water flow.

$$\nabla\vec{u} = 0$$

$$\frac{\partial\rho\vec{u}}{\partial t} + \nabla\cdot(\rho\vec{u}\vec{u}) - \nabla(\mu\nabla\vec{u}) = -\nabla p + \rho\vec{g} \qquad (1)$$

$$\phi_t + \vec{u}\nabla\phi = 0$$

where $\vec{u}$ – the velocity vector, t – time, p – the pressure, μ – the dynamic viscosity, ρ -the density.

Nondimensionalization of the system of equations (1) will be held as follows:

$$\vec{u}' = \frac{\vec{u}}{u},\ t' = \frac{t}{T},\ x' = \frac{x}{L},\ p' = \frac{p}{P},\ \vec{g}' = \frac{\vec{g}}{g},$$
$$(2)$$
$$\rho' = \frac{\rho}{\rho_\infty},\ \mu' = \frac{\mu}{\mu_\infty},\ \nabla' = \frac{1}{L}\nabla,\ \phi' = \frac{\phi}{\phi_\infty}$$

By substituting dimensionless quantities (2) into the initial system of equations (1) and discarding the strokes,we get a mathematical model for the two-phase flow:

$$\nabla \vec{u} = 0$$

$$\frac{\partial \vec{u}}{\partial t} + \vec{u} \cdot (\nabla \vec{u}) =$$

$$= -\frac{1}{\rho}\nabla p + \frac{1}{\rho \operatorname{Re}}\nabla(\mu \nabla \vec{u}) + \frac{1}{Fr^2}\vec{g} \qquad (3)$$

$$\phi_t + \vec{u}\nabla\phi = 0$$

$$\rho = \phi\rho_1 + (1-\phi)\rho_2$$

$$\mu = \phi\mu_1 + (1-\phi)\mu$$

where μ – the dynamic viscosity, ρ – the density, μ_1 – the dynamic viscosity of the air, μ_2 – the dynamic viscosity of the water, ρ_1 – the density of the air, ρ_2 – the density of the water, $Fr^2 = \frac{U^2}{gL}$ – the Froude number, $\operatorname{Re} = \frac{UL}{\nu} = \frac{UL\rho}{\mu}$ – the Reynolds number

Numerical algorithm

The splitting scheme of physical parameters is used for the numerical solution of the system (3) [20, 21]. In the first step, it is assumed that the transfer of momentum carried out only by convection and diffusion. The intermediate velocity field is computed using the Runge-Kutta method. In the second stage, the pressure field is calculated via the found intermediate velocity field. The Poisson equation for the pressure field is solved by the Jacobi method. In the third step, it is assumed that the transfer is carried out only by the pressure gradient. In the fourth phase, the phase equation is solved numerically by the Runge-Kutta method.

I. $\int_\Omega \frac{\vec{u}^* - \vec{u}^n}{\tau} d\Omega = -\oint_{\partial\Omega} (\nabla \vec{u}^n \vec{u}^* - \upsilon \Delta \vec{u}^*) n_i d\Gamma$

II. $\oint_{\partial\Omega} (\Delta p) d\Gamma = \int_\Omega \frac{\nabla \vec{u}^*}{\tau} d\Omega$

III. $\frac{\vec{u}^{n+1} - \vec{u}^*}{\tau} = -\nabla p.$

IV. $\int_\Omega \frac{\phi^{n+1} - \phi^n}{\tau} d\Omega = -\oint_{\partial\Omega} (\vec{u}^n \nabla \phi^{n+1}) n_i d\Gamma$

After all these four steps the density and the viscosity are calculated by formulas given the mathematical model.

Results of the numerical simulation

A domain with size of 3 to 1 is used for the numerical simulation of the two-dimensional system of equations. The number of cells is more than 50000. The height of the water level at the beginning is 0.6, the length is 1.1, the height of the dam is 0.1, Fr = 2, Re = 1000.

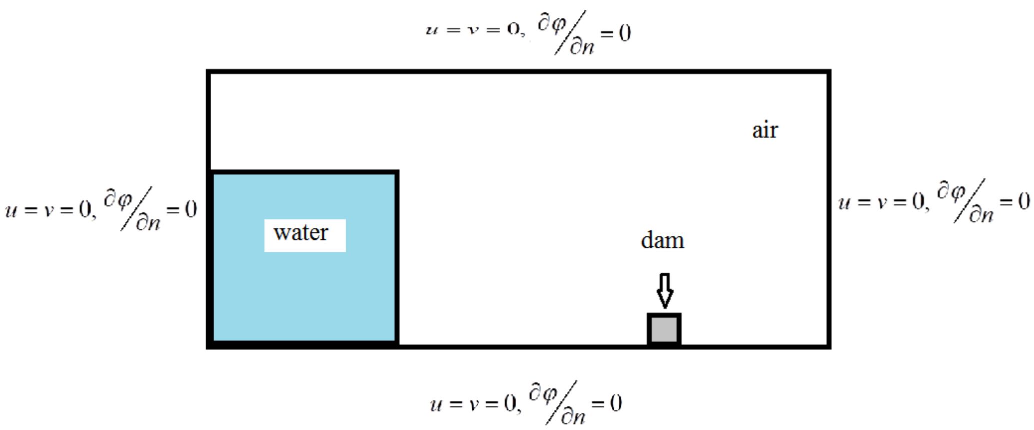

Figure 1 – The diagram of the boundary and initial conditions.

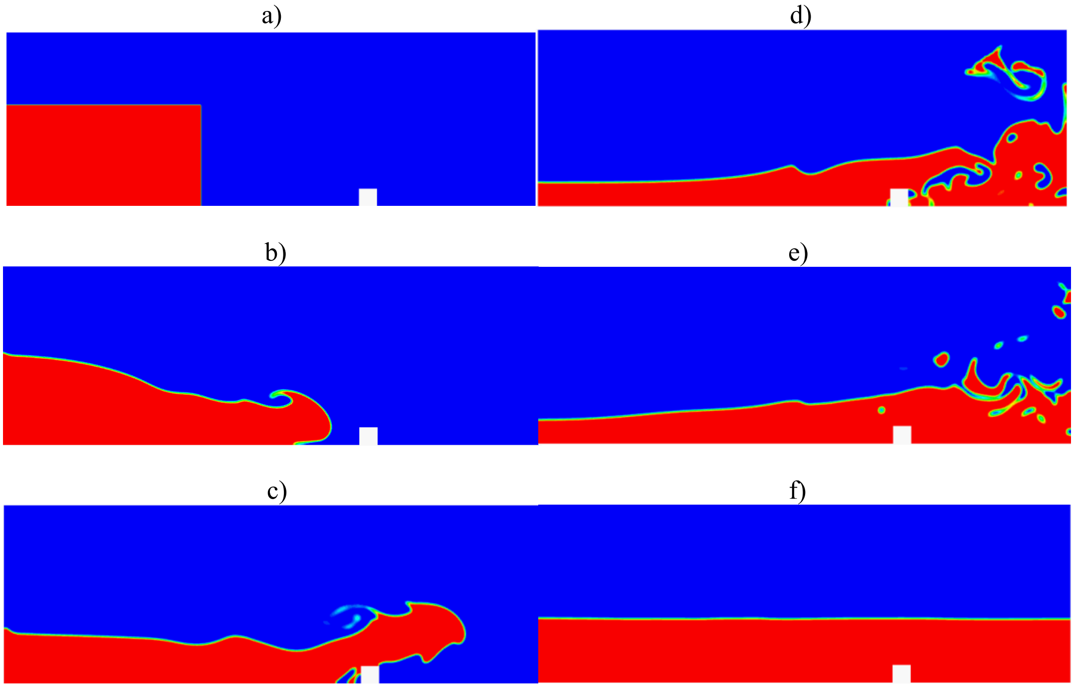

Figure 2 – Movement of the fluid flow with an obstacle in the form of a dam.

The boundary and initial conditions are set as shown in Figure 1 for the physical variables: the phase and velocity components. Figure 2 shows the stages of water motion pictures. Thus, at the initial time (t = 0), there is some water, which occupies the area of a rectangle with a height of 0.6 and 1.1 in length, enclosed by a glass wall on the right side. Further, the enclosure is removed, and the water movement begins in the direction of the dam. So at the picture b) at t = 0.1 water has not yet reached the dam.Further, after reaching the dam, provided that the water does not come in a stationary position, its movement is continued, and it starts to bend around an obstacle, in this case, the dam, as illustrated in Figures b) -f). Initially in this experiment a large amount of water is taken, that covers an area larger than the area of the reservoir, causing the reason why water goes around and bends the dam, which indicates that the height of the protective hydraulic engineering construction is not high enough to stop the water flow.

Conclusion

In carrying out this work, a two-phase mathematical model of fluid movement, based on the full two-dimensional Navier-Stokes equations and Kahn-Hilliard equation was numerically implemented. This mathematical model can be applied in the construction of new hydraulic structures, as well as in predicting of emergencies at already existing dams.

According to the obtained via the numerical studies data, one can say that in the construction of hydraulic structures one needs to take into account inadvance many factors such as the characteristics of the area and the characteristics of the dam in order to minimize the possibility of accidents in the future.

References

1. Biscarini C., Di Francesco S., Manciola P. CFD modeling approach for dam break flow studies // Hydrology and Earth System Sciences. – 2010. – Vol. 14. – P. 705-718.

2. LinaS.-Y., Yang F.-L., Chen C.-S., Hsieh S.-H. A parallel VOF IB pressure-correction method for simulation of multiphase flows // Applied Mathematical Modeling. – 2016. – Vol. 40. – P. 1800-1815.

3. Baker G.R., Meiron D.I., Orzsag S.A. Vortex simulations of the Rayleigh-Taylor instability // Physics of Fluids, – 1980. –Vol. 23. – P. 1485-1490.

4. Duchemin L., Josserand C., Calvin P. Asymptotic behavior of the Rayleigh-Taylor instability // Physical Review Letters. – 2005. – Vol. 94. – P. 224501-1 – 224501-4.

5. Verdon C.P., McCrory R.L., Morse R.L., Baker G.R., Meiron D.I., Orszag S.A. Nonlinear effects of multifrequency hydrodynamic instabilities on ablatively accelerated thin shells // Physics of Fluids. – 1982. – Vol. 25. – P. 1653-1674.

6. Popinet S., Zaleski S. A front-tracking algorithm for accurate representation of surface tension // International Journal for Numerical Methods in Fluids. – 1999. – Vol. 30. – P. 775-793.

7. Tryggvason G., Bunner B., Esmaeeli A., Juric D., Al-Rawahi N., Tauber W., Han J., Nas S., Jan Y.-J. A front-tracking method for the computations of multiphase flow // Journal of Computational Physics. – 2001. – Vol. 169. – P. 708-759.

8. Univerdi S.O., Tryggvason G. A front-tracking method for viscous, incompressible, multi-fluid flows // Journal of Computational Physics. – 1992. – Vol. 100. – P. 25-37.

9.Gerlach D., Tomar G., Biswas G., Durst F. Comparison of volume-of-fluid methods for surface-tension dominant two-phase flows // International Journal of Heat and Mass Transfer – 2006. – Vol. 49. – P. 740-754.

10. Hirt C.W., Nichols B.D. Volume of fluid (VOF) method for the dynamics of free boundaries // Journal of Computational Physics. – 1981. – Vol. 39. – P. 201-225.

11. He X., Chen S., Zhang R. A lattice Boltzmann scheme for incompressible multiphase flow and its application in simulation of Rayleigh-Taylor instability // Journal of Computational Physics. – 1999. – Vol. 152. – P. 642-663.

12.He X., Zhang R., Chen S., Doolen G.D. On the three-dimensional Rayleigh-Taylor instability // Physics of Fluids. – 1999. – Vol. 11. – P. 1143-1152.

13. Nie X., Qian Y.-H., Doolen G.D., Chen S. Lattice Boltzmann simulation of the two-dimensional Rayleigh-Taylor instability // Physical Review E 1998. – Vol. 58. – P. 6861-6864.

14. Chang Y.C., Hou T.Y., Merriman B., Osher S. Eulerian capturing methods based on a level set formulation for incompressible fluid interfaces // Journal of Computational Physics. – 1996. – Vol. 124. – P. 449-464.

15. Gomez P., Hernandez J., Lopez J. On the reinitialization procedure in a narrow-band locally refined level set method for interfacial flows // International Journal for Numerical Methods in Engineering. – 2005.– Vol. 63. – P. 1478-1512.

16. Sussman M., Smereka P., Osher S. A level set approach for computing solutions to incom-pressible two-phase flow // Journal of Computational Physics. – 1994. –Vol. 114. – P. 146-159.

17. Celani A., Mazzino A., Muratore-Ginanneschi P., Vozella L. Phase-field model for the Rayleigh-Taylor instability of immiscible fluids // Journal of Fluid Mechanics. – 2009. – Vol. 622. – P. 115-134.

18. Ding H., Spelt P.D.M., Shu C. Diffuse interface model for incompressible two-phase flows with large density ratios // Journal of Computational Physics. 2007.– Vol. 226. – P. 2078-2095.

19. Jacqmin D. Calculation of two-phase Navier-Stokes flows using phase-field modeling // Journal of Computational Physics. – 1999. – Vol. 155. – P. 96-127.

20. Anderson J.D. // Computational Fluid Dynamics the Basics with Applications. – 1995. – 547 p.

21. Chung T.J. Computational fluid dynamics. // CUP. – 2002.