

## NUMERICAL INVESTIGATIONS OF DAM-BREAK FLOW PROBLEM

S.N. Yakovenko<sup>1</sup>, E.E. Yakovenko<sup>1</sup>, K.C. Chang<sup>2</sup>

<sup>1</sup>*Khristianovich Institute of Theoretical and Applied Mechanics SB RAS,  
630090, Novosibirsk, Russia*

<sup>2</sup>*National Cheng Kung University, Tainan 70101, Taiwan, R.O.C.*

**Abstract.** A dam-break flow of water above the horizontal dry bed is studied numerically. Computational methods include different interface resolution techniques and the continuous surface force model to capture the surface tension effects. The results are compared with the measurement data. It is shown that the surface tension leads to the motion suppression and, thus, to acceleration decrease, in closer agreement with laboratory experiments.

The study is performed within the framework of the Integrated Basic Research Program of SB RAS for 2018-2020 (Project No. AAAA-A18-118021590030-8).

**Introduction.** The interface resolution techniques are of the key interest in many practical engineering applications. The main difficulty of these techniques is to produce quite sharp and realistic approximation of the interface between two immiscible fluids. The evident requirement of such a resolution is that the finite free-surface thickness predicted should be much smaller than the typical length scales of other physical and geometry parameters. On the other hand, one needs an efficient and stable solver to be applicable to practical situations.

One of the well-known and widely used methods to resolve the free surface is based on the concept of a fractional volume of fluid (VOF) [1]. It defines the function  $f$  of volume fluid fraction which is unity at points occupied by one (usually the denser) fluid and zero at points occupied by the other fluid. The method introduces an equation to compute the distribution of VOF function in which it corresponds physically to a step function with a zero-thickness discontinuity and numerically to a thin zone of sharp changes (the interface thickness is about a computational cell size). This  $f$  equation states that the VOF function moves with the fluid. The original VOF approach was applied in many studies, in particular, in predictions of dam-break flows [2] and Rayleigh–Taylor instability [3]. An alternative scheme to approximate the  $f$  fluxes is based on the MUSCL approach with high-order interpolants, e.g. the QUICK interpolants and the TVD limiter functions [4]. Different schemes for volume-fraction flux were compared in computations of a dam-break problem above the horizontal dry bed [2] chosen as a test case in many previous studies due to available experiment data [5].

The present computation involves also the surface tension effects according to the continuum surface force (CSF) model [6] applied for the Rayleigh–Taylor instability growth [3] too.

The studies include the thorough survey of preceding results of analytical and numerical investigations, laboratory experiments and field observations performed for related issues of solitary wave and dam-break wave propagation above the horizontal and sloping bottoms.

The developed numerical tools can be further applied to more general cases, e.g. to predict the dam-break bore above the sloped bottom [7].

**Governing equations.** Numerical simulation is based on the unsteady continuity, Navier–Stokes, volume fraction equations in Cartesian coordinates  $(x, y)$  for incompressible fluid where  $x$  and  $y$  are the horizontal and vertical coordinates,  $f = 1$  and  $f = 0$  are the volume-fraction values for denser and lighter fluids, respectively. The surface tension effects are introduced as the reformulated volumetric forces due to the CSF model [3, 6]. In the present study, the mollified volume fraction is taken to be equal to the volume fraction function defined numerically due to smoothing the solution by numerical scheme. One can also ignore the surface tension as in [2]

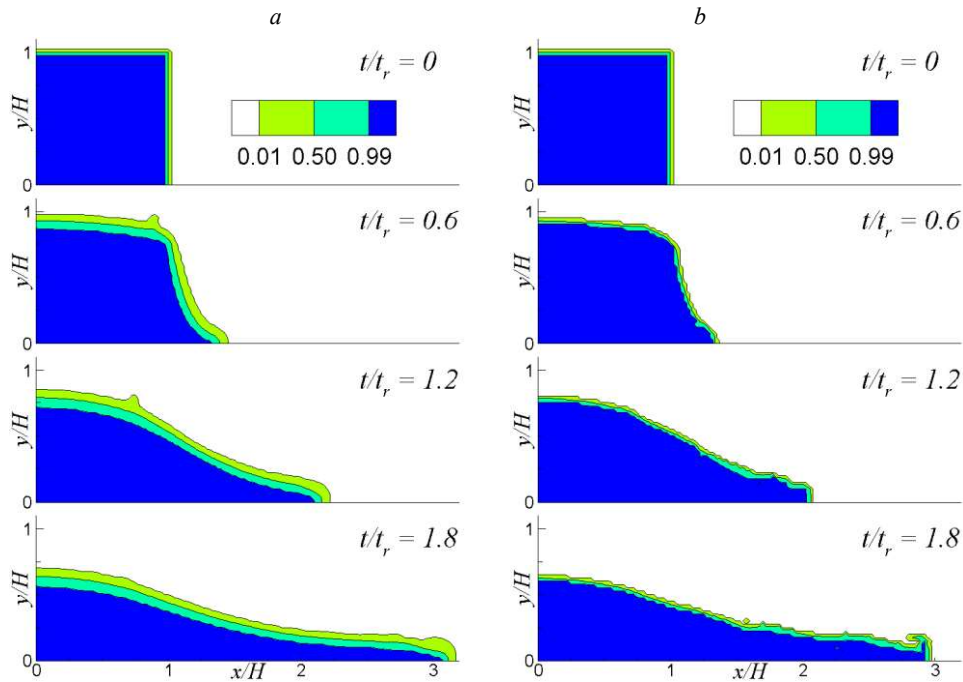


Fig. 1. Contour plots of water volume fraction obtained in computations with the CSF model for c4minmod (a) and VOF (b) advection schemes in the volume-fraction equation (Cases 3 and 4).

and compare the results. Discretization of equations is done on a staggered grid to prevent mismatch between the velocity and pressure fields. Other numerics details are given in [2, 3].

**Computation results.** The runs are made without CSF model (Cases 1, 2) and with CSF model (Cases 3, 4), using two selected advection schemes described in [2]. Case 1 taken from [2] and Case 3 (both denoted as “c4minmod”) correspond to the TVD MUSCL scheme with QUICK interpolants and the compressive minmod TVD-limiters [3] for advection fluxes in the  $U$ ,  $V$ ,  $f$  equations. Case 2 taken from [2] and Case 4 (both denoted as VOF) replace the advection scheme in the  $f$  equation by the upwind-downwind donor-acceptor VOF procedure [1] while keeping the same flux expression (as in c4minmod approach) for the  $U$  and  $V$  equations.

From the comparison of different scheme performance (fig. 1) one may reconclude [2] that the VOF method (Case 4) gives the sharper free surface whereas the c4minmod scheme (Case 3) produces the larger interface smearing. However, some distortion of volume-fraction contours is seen for VOF (and also for c4minmod Cases at more refined grids in [2]). The distortion of the free surface (corresponding to the contour  $f = 0.5$ ) and its wave-like behavior may be caused not only by numerical scheme drawback but also by physical phenomena like the developing turbulence. For the VOF scheme, more realistic reproduction of the broken dam problem evolution can be achieved using closer levels of effective interface boundaries ( $f = 0.33$  and  $0.67$ ), and some shortcomings can be avoided at smaller time step [2].

For the water surge front position and column height (fig. 3), a small delay of experimental values from predicted ones at  $t < 2t_r$  can be explained by the measurement conditions [5]. In the experiments, the initial water column was constrained by a very thin waxed paper diaphragm, and a heavy current from car batteries was used to free the waxed paper and thus allow motion to begin. The value was taken as the time when a heater current was applied, and the water motion began with a little unfixed delay in the time interval  $0 < t < 0.2t_r$  [5].

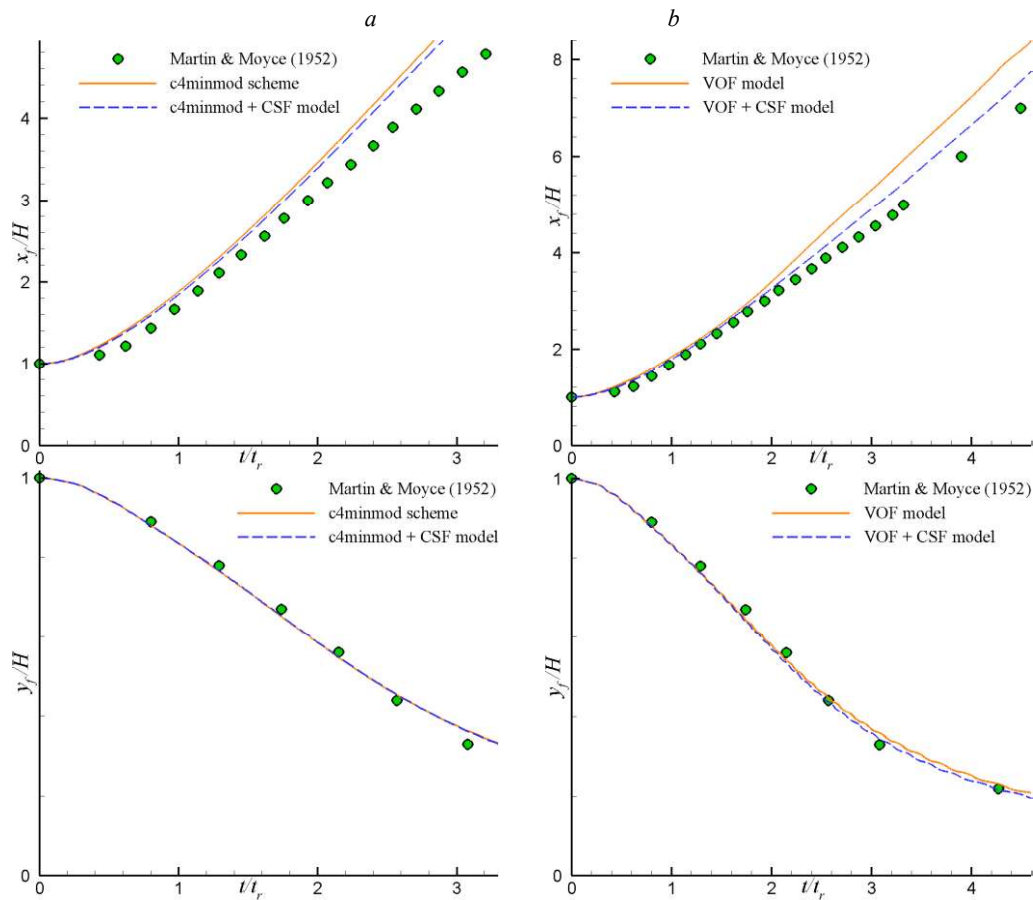


Fig. 2. Surge front position  $x_f$  and column height  $y_f$  versus time with and without CSF model, for c4minmod (a) and VOF (b) schemes of convection fluxes in the volume-fraction equation.

#### REFERENCES

1. Hirt C.W., Nichols B.D. Volume of fluid (VOF) method for the dynamics of free boundaries // J. Comp. Phys. 1981. Vol. 39. P. 201-225.
2. Yakovenko S.N., Chang K.C. Volume fraction flux approximation in a two-fluid flow // Thermophysics and Aeromechanics. 2008. Vol. 15. P. 169-186.
3. Yakovenko S.N. The effects of density difference and surface tension on the development of Rayleigh–Taylor instability of an interface between fluid media // Fluid Dynamics. 2014. Vol. 49, № 6. P. 748-760.
4. Kelecy F.J., Pletcher R.H. The development of a free surface capturing approach for multi-dimensional free surface flows in closed containers // J. Comp. Phys. 1997. Vol. 138. P. 939-980.
5. Martin J.C., Moyce W.J. An experimental study of the collapse of liquid columns on a rigid horizontal plane // Phil. Trans. Royal Soc. of London, Ser. A. 1952. Vol. 224. P. 312-324.
6. Brackbill J.U., Kothe D.B., Zemach C. A continuum method for modelling surface tension // J. Comput. Phys., 1992. Vol. 100, Is. 2. P. 335-354.
7. Lin C., Kao M.J., Wong W.Y., Shao Y.P., Fu C.F., Yuan J.M., Raikar R.V. Effect of leading waves on velocity distribution of undular bore traveling over sloping bottom // European Journal of Mechanics B. Fluids. 2019. Vol. 73. P. 75-99.