Studies of free surface evolution in a dam-break flow above horizontal bottom

S N Yakovenko\(^1\) and K C Chang\(^2\)

\(^1\)Khristianovich Institute of Theoretical and Applied Mechanics SB RAS, Novosibirsk, Russia
\(^2\)National Cheng Kung University, Tainan, Taiwan, R.O.C.

E-mail: yakovenk@itam.nsc.ru

Abstract. A dam-break flow of water above horizontal bottom 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 surface tension leads to the motion suppression and, thus, to acceleration decrease, in closer agreement with laboratory experiments.

1. 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 (usually, liquid and gas). 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 basic advection method proposed in the VOF technique is a special donor-acceptor procedure involving downwind or upwind fluxes depending on interface orientation. 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 study involves also the surface tension effects according to the CSF model \([6]\) applied to compute the Rayleigh–Taylor instability evolution \([3]\) too.
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].

2. Mathematical model

Numerical simulation is based on the unsteady continuity, Navier–Stokes and volume fraction equations in Cartesian coordinates \((x, y)\) for incompressible fluid:

\[
\frac{\partial U}{\partial t} + \frac{\partial V}{\partial y} = 0
\]

\[
\frac{\partial U}{\partial t} + \frac{\partial U^2}{\partial x} + \frac{\partial UV}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \frac{1}{\rho} \frac{\partial}{\partial x} \left( \eta \frac{\partial U}{\partial x} \right) + \frac{1}{\rho} \frac{\partial}{\partial y} \left( \eta \frac{\partial U}{\partial y} \right) + F_U^s
\]

\[
\frac{\partial V}{\partial t} + \frac{\partial UV}{\partial x} + \frac{\partial V^2}{\partial y} = -\frac{1}{\rho} \frac{\partial p}{\partial y} + \frac{1}{\rho} \frac{\partial}{\partial x} \left( \eta \frac{\partial V}{\partial x} \right) + \frac{1}{\rho} \frac{\partial}{\partial y} \left( \eta \frac{\partial V}{\partial y} \right) - g + F_V^s
\]

\[
\frac{\partial f}{\partial t} + \frac{\partial (Uf)}{\partial x} + \frac{\partial (Vf)}{\partial y} = 0
\]

where \(U\) and \(V\) are the horizontal and vertical components of the velocity vector, respectively, \(x\) and \(y\) are the horizontal and vertical coordinates, respectively, \(g\) is gravity acceleration, \(\rho\) is density, \(p\) is pressure, \(t\) is time, \(\rho_1 = \rho f + \rho_2 (1 - f)\), \(\eta_1 = \eta f + \eta_2 (1 - f)\). The values \(\rho_1\) and \(\eta_1\) correspond to denser fluid (liquid) at \(f = 1\), whereas \(\rho_2\) and \(\eta_2\) correspond to lighter fluid (gas) at \(f = 0\).

In (2)-(3), the surface tension effects are introduced as the reformulated volumetric forces due to the continuum surface force (CSF) model written in [3, 6] where \(F_i^s = (\sigma \kappa / \rho) \left( \frac{\partial f}{\partial x} / \partial x \right)\), with the surface curvature \(\kappa = -\hat{n}_i / \partial x\), the unit normal \(\hat{n}_i = \partial f / \partial x\) to the surface, and the surface tension coefficient \(\sigma\). In the present study, the mollified volume fraction \(\tilde{f}\) is taken to be equal to the volume fraction function defined numerically \((\tilde{f} = f)\) due to smoothing the solution by numerical scheme. It is possible also to ignore the surface-tension effects by \(\sigma = 0\) as in [2] and compare the results.

Discretization of equations is carried out on a staggered grid to prevent mismatch between the velocity and pressure fields. All details of the numerical solution are given in [2, 3].

3. Computation results

The initial state of the sudden collapse of rectangular water column is schematically shown in figure 1.

Computation conditions follow those in [2, 5]: \(H = 0.05715\) m, \(g = 9.80665\) m/s\(^2\), \(\rho_1 = 998\) kg/m\(^3\), \(\rho_2 = 1.19\) kg/m\(^3\), \(\eta_1 = 0.99 \times 10^{-3}\) kg/(m·s), \(\eta_2 = 1.84 \times 10^{-3}\) kg/(m·s). The surface tension coefficient is \(\sigma = 0.072\) N/m. The reference quantities \(L_s = H\), \(U_s = \sqrt{gH}\), \(t_s = \sqrt{H/g}\), \(\rho_1 = \rho_1\), \(\eta_1 = \eta_h\) are used for non-dimensionalization, producing the reference Reynolds number \(Re_c = \rho U_s L_s / \eta_h = 43130\).

The computations 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. Note, both the first-order upwind scheme and the QUICK scheme without TVD limiters are not applied for flux approximation in (2)-(4). The former produces the thick unrealistic interface due to numerical diffusion, and the latter leads to significant distortion of volume-fraction contours due to spurious oscillation features of the QUICK scheme [2].
$L_x$

$y$

$H$

$\text{air (} \rho_2, \nu_2 \text{)}$

$H$

$L_y$

$\text{water (} \rho_1, \nu_1 \text{)}$

$(t = 0^+)$

$\vec{g} = (0, -g)$

**Figure 1.** Scheme of dam-break problem (initial state).

**Figure 2.** 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).
From the comparison of different scheme performance (figure 2) one may re-conclude [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 $f = 0.5$ contour) 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 (figure 3), a small delay of experimental values from predicted ones at $t < 2r_t$ 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 $t = 0$ 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 $0 < t < 0.2r_t$ [5].

Figure 3. 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.
A reason of increasing $x_f$ over-estimation at $t > 2t_r$ (observed also in [4]) for all schemes was not evident in [2]. However, it was concluded [3] that similar actions of both viscosity and surface tension effects lead to the motion suppression and, thus, to acceleration decrease. This effect is clearly seen in figure 3 where values of $x_f(t)$ are smaller for both schemes and closer to experiment points when the surface tension is taken into account. Moreover, this improving effect is stronger (figure 3b) for the VOF method giving the sharper free surface, and the smallest one for the scheme (figure 3a) producing the thicker interface. Comparison also shows approximate coincidence of column height $y_f(t)$ for all schemes except the VOF + CSF model which gives small improvement at large times.

Acknowledgments
The study is made partly within the framework of the Program of Fundamental Scientific Research of the state academies of sciences in 2013-2020 (Project No. AAAA-A17-117030610128-8) and Integrated Basic Research Program II.1 of SB RAS (Project No. 2).

References
[1] Hirt C W and Nichols B D 1981 J. Comp. Phys. 39 201
[2] Yakovenko S N and Chang K C 2008 Thermophys. Aeromech. 15 169
[3] Yakovenko S N and Chang K C 2014 Fluid Dynamics 49 748
[4] Kelecy F J and Pletcher R H 1997 J. Comp. Phys. 138 939
[5] Martin J C and Moyce W J 1952 Phil. Trans. Royal Soc. of London, Ser. A 224 312
[6] Brackbill J U, Kothe D B, and Zemach C 1992 J. Comp. Phys. 100 335
[7] Lin C, Kao M J, Wong W Y, Shao Y P, Fu C F, Yuan J M and Raikar R V 2019 Europ. J. Mech. B/Fluids 73 75