Numerical investigations of dam-break flow problem

S N Yakovenko¹, E E Yakovenko¹ and K C Chang²

¹Khristianovich Institute of Theoretical and Applied Mechanics, SB RAS, 4/1 Institurskaya st., Novosibirsk 630090, Russia
²National Cheng Kung University, Tainan 70101, Taiwan, R.O.C.

s.yakovenko@mail.ru

Abstract. A dam-break flow of water above horizontal dry bottom is studied numerically, after the comprehensive review of available literature. Computation tools include different interface resolution techniques and the continuous surface tension force model. The results are compared with the measurement data. It is shown that surface tension leads to the motion suppression and, therefore, to the acceleration decrease, providing closer agreement with laboratory experiments.

1. Introduction

One of the urgent issues of fluid dynamics of technical and natural systems is the study of evolution of the flow with a free surface near the coastline. Oncoming perturbations with huge water masses caused by natural and industrial disasters (tsunamis, tidal waves, bores produced by a sudden dam breaking, storm surf) can lead to serious consequences. In 2004 and 2011, large-scale natural disasters related to the tsunami occurred. When the waves move to the coastal zone, their height and energy increase significantly. The wave breaking near the coast is followed by run-up and run-down of large water masses, which can form a strong front and result in the coastal infrastructure damage. The velocity field reproduction at all stages of the flow evolution is a key factor for developing activities to mitigate consequences and would be useful to further understand various aspects of the impact of water masses on a coastline: land flooding, coastal erosion, wave structure interactions, sediment transport, morphology.

The water movement near the coast occurs in the zones of surf (located between the wave breaking and maximum run-down positions) and splash (between the wave run-up and run-down maxima). Study of the free surface perturbation evolution in a reservoir over horizontal and inclined bottoms for models of bore and solitary wave reflects a real picture of emergence of waves on the coast. Bore is observed in the splash zone near the coast, in an estuary (a funnel-shaped mouth of a river) and is supposed to be a moving discontinuity, where the height of the free surface grows from the level of still water to the peak of wave motion, and the horizontal velocity varies from zero to a noticeable value. Bore is usually modeled by a sudden rising of a fence (dam break), which generates a dam break flow towards a lower level of the free surface. Bore observed in tsunamis and leading to land flooding can be also viewed as a combination of incoming solitary waves with different heights and time intervals between them: the movement of bore to a region with decreasing depth generates a number of solitons.

In the present study, to resolve the free surface, the VOF concept [1] is used as in many studies, for instance, in prediction of dam-break flow [2] and Rayleigh–Taylor instability (RTI) [3]. An alternative
scheme to approximate the volume fraction fluxes is based on the MUSCL approach with the QUICK interpolants and TVD limiters [4]. A dam-break flow over horizontal dry bed is chosen as a canonical test here and in many preceding studies due to available experiments data [5, 6]. The surface tension effects are inserted via the CSF model [7] used to compute RTI [3] too. The developed tools can be further applied to more general cases, e.g. to predict the dam-break bore above the sloped bottom [8].

2. Literature survey
Because of the limited accuracy of measurements, the experimental data was very scarce in early works, and simplified analytical approaches with the depth-averaged (shallow water) equations were considered. The recent development of the non-intrusive measurement tools (ADP, LDV, PIV) allowed one to obtain data for the structure of waves and bores in the channel and field observations (e.g. [8-11]).

A solitary wave moving onto an inclined shore was studied in many papers, where the free surface evolution, the breaking type, the external flow velocity field, and the maximum run-up height were considered. Recently, the velocity field evolution in the incoming wave prior to breaking, with subsequent run-up and run-down, has been studied [10, 11]. The analysis of evolution stages was carried out, data are obtained for various bottom slopes and the ratios of wave height to water depth.

The dam-break bore was studied for a number of tests, including the horizontal dry (a) and wet (b) beds, sloped bottom (c), smooth or rough surfaces, obstacles (wavebreaker, dyke, reef, prism, etc.).

In the first measurements [5] for test (a), the evolutions of free surface, (positive) surge front location \( x_1 \) and water column height \( y_1 \) have been obtained for five cases of square \((H×H)\) and rectangular initial water areas of depths between 57 and 115 mm. More recent experiments [6] in the larger-size channel with initial water region of depth 0.3 m, length 3.5 m also studied the evolution of the negative wave front and flow velocities. Such a flow was first computed by the VOF method in [1] to yield proper result for \( x_1(t) \) with refined uniform grid of cell size \( H/40 \). Later numerical studies have shown that 3D effects of lateral walls can give slight improvement for \( x_1(t) \) [4], as well as adding of extra turbulent viscosity in 2D runs based on RANS or LES-like techniques [12, 13]. Different volume-fraction-flux schemes and wall boundary conditions were compared in simulations [2].

For the test (b), by lifting the gate in the flume, which initially separated high and low water levels, the experiment [14] was made for weak and strong bores with turbulence generation, using the flow visualization and the water sensors to obtain the free surface distributions and to measure the wave propagation velocities. The similar studies in the water flume [15] for initial stages of dam-break flows over dry and wet beds revealed the mushroom-like jets occurring at small times which is captured in simulations by the Navier–Stokes equations [12, 16] but not by the shallow-water equations [15, 16].

For test (c), turbulence models when using with depth averaging are also not able to describe local flow features, for example, turbulence during bore breaking, due to neglect of noticeable vertical flow variations [17]. In experiments [18], bore types (strong and weak) are classified depending on the ratio of the initial water levels in front of the barrier and behind it. The effects of bore breaking interacting with a wedge of water in front of it, pushing the wedge to the shore, which is very different from the behavior described by the shallow water equations, are shown. In [17], the dam break bore, moving on the constant depth water, then over the 1:8 slope, has been studied based on the VOF and RANS methods, with the \( k-ε \) model. Free surface distributions, averaged velocity, turbulence energy during shoaling, breaking, run-up and run-down, for strong and weak bores are obtained where small-scale effects, in particular, mini-collapse during run-up and returned bore during water run-down, are found.

3. Mathematical model
The present simulation is based on the unsteady continuity, Navier–Stokes, volume fraction (\( f \)) 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 effect is inserted as the reformulated volumetric force due to the CSF model [3, 7]. The mollified volume fraction is taken here 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] and compare 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].

4. Computation results
The initial flowfield configuration for the problem of the rectangular water column sudden collapse is shown schematically in figure 1.

![Figure 1. Scheme of dam-break problem (initial state).](image)

Parameters of numerical simulations are as in [2, 5]: \( H = 0.05715 \) m, \( g = 9.81 \) m/s\(^2\), \( \rho_1 = 998 \) kg/m\(^3\), \( \rho_2 = 1.19 \) kg/m\(^3\), \( \eta_1 = 0.99\cdot10^{-3} \) kg/(m\(\cdot\)s), \( \eta_2 = 1.84\cdot10^{-5} \) kg/(m\(\cdot\)s). The surface tension coefficient value is \( \sigma = 0.072 \) N/m. The reference Reynolds number \( \text{Re}_r = \rho_r U_r L_r / \eta_r \) = 43130 is based on the reference quantities \( L_r = H, U_r = (gH)^{1/2}, \tau_r = (H/g)^{1/2}, \rho_r = \rho_1, \eta_r = \eta_1 \) used for non-dimensionalization.

The computations are performed without the CSF model (Cases 1, 2) and with the 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 [4] 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 that 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].

The comparison of results for different schemes (figure 2) shows as in [2] that the VOF method gives the sharper free surface whereas the c4minmod scheme produces the larger interface smearing. However, some distortion of contours of volume fraction \( f \) is seen for VOF (and for c4minmod Cases at more refined grids in [2] too). The distortion of the free surface (given by the contour of \( f = 0.5 \)) and its wave-like behavior may be caused not only by numerical scheme drawback but also by specific 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 positive surge front position and the water column height (figure 3), a small delay of the experimental values from the predicted ones at \( t < 2t_r \) can evidently be explained by the measurement conditions [5]. In the experiments, the initial water column was constrained by a thin waxed paper
diaphragm, and a heavy current from car batteries was used to rupture 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 \( 0 < t < 0.2r_c \) [5].

**Figure 2.** Contour plots of the 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).

**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 viscosity and surface tension effects lead to the motion suppression and, thus, to acceleration decrease, as in [4, 12, 13] due to effects of lateral walls or extra turbulent or sub-grid scale viscosities. The damping 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 correcting 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 water column height $y_f(t)$ for all schemes except the VOF + CSF model giving small improvements at large times.

Acknowledgments
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).

References
[1] Hirt C W and Nichols B D 1981 J. Comp. Phys. 39 201
[2] Yakovenko S N and Chang K C 2008 Thermophysics and Aeromechanics 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] Lauber G and Hager W H 1998 J. Hydraul. Res. 36 291
[7] Brackbill J U, Kothe D B and Zemach C 1992 J. Comp. Phys. 100 335
[8] Lin C, Kao M J, Wong W Y, Shao Y P, Fu C F, Yuan J M and Raikar R V 2019 European Journal of Mechanics B. Fluids 73 75
[9] Raubenheimer B 2002 J. Geophys. Res. 107 C11
[10] Lin C, Yeh P H, Kao M J, Yu M H, Hsieh S C, Chang S C, Wu T R and Tsai C P 2015 J. Waterway, Port, Coastal, Ocean Eng. 141 04014038
[11] Lin C, Kao M J, Tzeng G W, Wong W Y, Yang J, Raikar R, Wu T R and Liu P L F 2015 J. Earthquake and Tsunami 9 154002
[12] Shigematsu T, Liu P L F and Oda K 2004 J. Hydraul. Res. 42 183
[13] Hieu P D, Katsuoshi T and Ca V T 2004 Appl. Math. Modelling. 28 983
[14] Quecedo M, Pastor M, Herreros M I, Fernandez Merodo J A and Zhang Q 2005 Comput. Methods Appl. Mech. Engng. 194 3985
[15] Yeh H H and Mok K M 1990 Phys. Fluids A 2 821
[16] Stanisby P K, Chegini A and Barnes T C D 1998 J. Fluid Mech. 374 407
[17] Zhang Q and Liu P L F 2008 Coast. Eng. 55 1113
[18] Yeh H H, Ghazali A and Marton I 1989 J. Fluid Mech. 206 563