B08

Two-Dimensional Hydrodynamic Modeling of Air-Water 2-Phase Flow in Urban Sewer System

〇Chong Soo HAN, Kenji KAWAIKE, Hajime NAKAGAWA

Urban inundation disasters due to increasing of torrential local rainfall and decreasing of impermeable area cause serious problem in many countries. To mitigate the damage of urban inundation, various structural strategies have been carried out, one of which is an underground drainage pipe system. Deeply underground, especially development area in mega city, drainage pipe system with huge-diameter has been laid in recent years. In order to estimate the effect of pipe system, diversion flow rate must be estimated precisely.

However, when the water flows into the pipes, the air will be taken into the flow as well which makes it difficult to estimate the flow discharge. This kind of flow containing air-mass or bubbles is referred as 2-phase flow is such that both open channel flow and pressurized flow are presented in one systems. In a drainage pipe shows the different hydraulic characteristics than usual open channel flow and the compressed air in domain pipes.

Numerical simulations of multi-phase flow where all continuum time and length scales are fully resolved, have had a particularly important impact. Such simulations have long been a standard tools for studies of turbulent single phase flow, but for multiphase flows, where two or more immiscible fluids separated by a sharp interface occupy the same domain, progress has been considerably slower. Nevertheless such simulations are gradually becoming more common.

The existing mixed-flow models typically solve the 2-dimensional Shallow Water Equation in combination with the 1-dimensional surge equation for the sewer-pipe and a separate set of 1-dimensional continuity and momentum equations for the vertical pipe. Problem could rise when shallow water equation is applied to violent flow field. There is significantly different in pressure distribution. Not all the existing mixed-flow models solve the flows in the vertical pipe, so simulation is greatly simplified. And the existing 2-phase numerical models developed for mixed-flows treat the water and air phase as a rigid column with no details resolved within the columns. The interaction of the 2-phase is often simplified by 1-dimensional momentum equation. The motions of an air-water mixture in the pipe have not be part of any of the existing models. The shape of air pocket are very different and often irregular in flow transitions. The simulation of air escape in the vertical pipe was not conducted by a fully dynamic model. A few attempts have been reported simulating the upward air pocket rising motion, but it is greatly simplify the physics. And the air pocket is typically treated as a slug flow, The momentum exchange only occurs at top and bottom of vertical pipe.

Computations of 2-phase flows go back to the origin of computational fluid dynamics. The Marker and Cell(MAC) method was designed specifically for free-surface and multiphase flows and used to simulate interfacial instabilities, gravity current, waves and droplet collisions with walls, and other problems. The majority of numerical methods currently used for detailed simulation of multiphase flows are based on the so-called one-field formulations of the Navier-Stokes equations, where one set of equations, describing the flow of all the fluids involved. The equations are usually solved on a regular structured grid, using a second order projection method where the solution is first updated without accounting for the pressure, the pressure is found from the divergence of the temporary solution, and the initial velocity is then corrected by adding the gradient of the pressure.

In this paper, I will consider the unsteady motion of two incompressible and immiscible fluid-air where the location of the interface separating the different 2-phase will change with time. In some cases one fluid is immersed in the other as a disperse phase consisting of bubbles or drops but in other cases the interface is more complex and it is difficult to identify the disperse phase. For simplicity I will focus our attention on disperse flows, often using the motion of only one bubble or a drop as the test case.

First, I conducted a very simple Navier-Stokes solver for variable density flow assuming that the viscosity of both are the same and ignore surface tension. To solve the density advection, I used a simple upwind method. Computational algorithm is as following. 1) Find a temporary velocity using the advection and the diffusion terms only. 2) Find the pressure needed to make the velocity field incompressible. 3) Correct the velocity by adding the pressure gradient.

And then, instead of advecting the density directly, we can move the interface and reconstruct the density from its location. This is usually called front-tracking. Unlike when we advect the density directly, using front tracking is a two-step process: move the boundary and then construct the density field. In some cases we define an indicator function that is used to set the density and other material properties but here we use the density as the indicator function. Generally the interfaces must be dynamically restructured by adding and deleting points as the interface deforms. In three dimension the correct data structure can be critical for the successful implementation of the method. For 2-dimensional flow, on the other hand, essentially any data structure can be made to work rather easily.

And we make more complete by allowing for different viscosities and by introducing surface tension. We also make the time integration second order and allow for the possibility of subtracting the hydrostatic pressure gradient. In simulations of immiscible multiphase flows we are frequently interested in length scales where surface tension is important. Surface tension acts only at the interface. For calculating the different we must use the full deformation tensor when computing the viscous stresses.

But for the more complicated simulation, we have to think how to conduct natural things exactly. Using this simulation model, examine a wide variety of problems although only for 2-dimensional flow. There is more changes that fist of all allow us to use the grid points a little more economically and to more easily simulate higher Reynolds number flows.

References

Finite Volume-based Volume-of-Fluid Method for the Simulation of Two-Phase Flows in Small Rectangular Channels, B. E. Ghidersa, 2004

A compact numerical implementation for solving Stokes equations using matrix-vector operations, Tao Zhang, Amgad Salama, Shuyu Sun, Hua Zhong, Procedia Computer Science, 2015 (51)

A Font-Tracking Method for the Computations of Multiphase Flow, G. Rtyggvason, B.Bunner, A.Esmaeeli, D. Juric, N.Al-Rawahi, W. Tauber, J. Han, S.Nas and Y-J. Jan, Journal of Computational Physics, 2001(169)

Development of Numerical Model Concerning the Behavior of Water and Air in a Long Inverted Siphon – A proposal of Unsteady Numerical Model Based on Hydraulic Experiment Model-, Khotato KUSHIYANA, Seiji KAMIOKA, Tadashi YAMADA, J.Japan Soc.Hydrol.& Water Resour., 2003(16)