NEW TRENDS IN FLUID MECHANICS RESEARCH
Proceeding of the Fifth International Conference on Fluid Mechanics, *Aug.15-19, 2007, Shanghai, China*

Development of a 3D Free Surface Capturing Code for Coastal Engineering Flow Problems

L. Qian*, D. M. Causon , C. Mingham

Centre for Mathematic Modelling and Flow Analysis, The Manchester Metropolitan University, Manchester, M1 5GD, England, UK

Email: L.Qian@mmu.ac.uk

Abstract A free surface capturing code has been developed for 3D hydraulic and coastal engineering flow problems. The underlying scheme is based on the solution of the incompressible Navier-Stokes equations for a variable density fluid system. The free surface is captured automatically during the calculation using a time-accurate artificial compressibility method and high resolution Godunov-type Riemann solver based algorithm. The accuracy of the code is being validated against some experimental results.

Key words: free surface, approximate Riemann solvers, artificial compressibility method, Cartesian cut cell method

INTRODUCTION

Numerical simulation of 3D flow problems with a moving free surface separating two or more immiscible fluids, which may involve interface break-up and/or reconnection, or entrapment of one fluid into the other, are complicated and difficult, but of huge significance to many engineering sectors such as aerospace, civil and process engineering. The situation is further complicated if the boundaries are in motion or moving bodies are present in the fluid system.

During the last twenty years or so, a number of numerical techniques have been put forward for modelling free surface problems and most of them can be put into two broad categories: surface-tracking methods and surface capturing methods, and others can be categorised as a hybrid of the two basic methods. In surface tracking methods, the free surface is treated as a sharp interface whose motion is followed explicitly. Early work of surface-tracking methods [1] solves the flow equations in the liquid region only with the free surface treated as a moving boundary of the computational domain. More recently this method has been extended to solve both fluids either separately or on a single set of governing equations for both 2D and 3D flows [2]. In this method, a separate mesh (lines for 2D and surfaces for 3D) of lower order is employed to represent the free surface. As the front moves, it deforms and stretches and the resolution of some parts of the front can become inadequate, while other parts become crowded with front elements. To maintain accuracy, either additional elements (grids) must be added or the points must be redistributed to maintain adequate resolution. Furthermore, it is not an easy task for this method to cope with topological changes such as the breaking and merging of the interfaces. On the other hand, surface capturing methods, including, among others, volume-of-fluid (VoF) based method and level set method offered credible alternatives to the surface-tracking methods due to its intrinsic properties in handling with ease the topological changes at interfaces. In these methods, both fluid regions can be solved on a fixed grid system with the free surface identified by a marker function such as the volume fraction in the VoF method [3] or the zero level set of a smooth function, e.g. the signed distance from the interface as in the level set method [4]. Early work of VoF method consists of two parts: an interface reconstruction algorithm using simple line interface calculation (SLIC) or by various piecewise linear interface calculations (PLIC) for approximating the interface from the set of volume fractions, and a VoF transport algorithm for determining the volume fractions at the new time level from the velocity field and the reconstructed front. While the explicit reconstruction can define sharp interfaces, this process can lead to small pieces of fluid being non-physically ejected as flotsam or jetsam and also its extension to 3D is complicated. Recent versions of VoF methods such as that of Lafaurie et al [5] or Ubbink [6], have abandoned the explicit interface reconstruction phase, instead treating the free surface as a transitional layer between the two fluids. Special complex solvers independent of the main flow solver are used to maintain the sharpness of the interface. A similar but more consistent approach has been proposed initially by Kelecy and Pletcher [7] and Pan et al [8], based on the artificial compressibility method with firm foundations of high resolution Riemann solvers for discontinuity. In this method, the free surface is treated as a contact discontinuity in the density field or via volume fraction function in a manner similar to shock-capturing in compressible flows. The method is robust and simple and it has since been adopted by the research group in conjunction with a Cartesian cut cell grid generation method for 2D flow problems with both free surfaces and moving bodies [9∼11].

The current study is mainly concerned with the extension of the 2D code to 3D flow problems. In the following sections, the mathematical formulation including its numerical implementation of the method is briefly reviewed. The results for a test case used to validate the code will be discussed. Other results relevant to real coastal engineering flow problems including the implementation of the Cartesian cut cell method for complex solid geometries and moving bodies will also be presented in the conference.

NUMERICAL METHOD

The integral form of the three-dimensional incompressible Euler equations for a fluid system with a variable density field can be written as

$$\frac{\partial}{\partial t}\iint\limits_{\Omega}\boldsymbol{Q}\,\partial\Omega+\oint\limits_{S}\boldsymbol{F}\cdot\boldsymbol{n}\,\partial S=\iint\limits_{\Omega}\boldsymbol{B}\,\partial\Omega \tag{1}$$

where Ω is the domain of control volume, S is the boundary surrounding Ω, $\boldsymbol{n}$ is the unit normal to S in the outward direction, $\boldsymbol{Q}$ is the vector of conserved variables, $\boldsymbol{F}$ is the vector of flux function through S and $\boldsymbol{B}$ is the source term for body forces. For three dimensional problems, by using the artificial compressibility method i.e. by introducing a fictitious time derivative of pressure into the constraint equation for incompressibility and assuming the only body force is the gravity, $\boldsymbol{Q}$, $\boldsymbol{F}$, and $\boldsymbol{B}$ are given as

$$\boldsymbol{Q}=\begin{bmatrix}\rho\\ \rho u\\ \rho v\\ \rho w\\ p/\beta\end{bmatrix},\ \boldsymbol{F}=\begin{bmatrix}\rho u\\ \rho u^2+p\\ \rho uv\\ \rho uw\\ u\end{bmatrix}\vec{i}+\begin{bmatrix}\rho v\\ \rho uv\\ \rho v^2+p\\ \rho vw\\ v\end{bmatrix}\vec{j}+\begin{bmatrix}\rho w\\ \rho uw\\ \rho vw\\ \rho w^2+p\\ w\end{bmatrix}\vec{k},\ \boldsymbol{B}=\begin{bmatrix}0\\ 0\\ -\rho g\\ 0\\ 0\end{bmatrix} \tag{2}$$

where u, v and w are the flow velocity components, ρ is the density, p is the pressure, β is the coefficient of artificial compressibility and g is the gravitational acceleration. Introducing a time derivative of pressure into the continuity equation produces a system of hyperbolic equations which can then be solved by any of the recently developed upwind finite volume techniques such as the characteristics based Godunov-type schemes. Clearly, from this formulation, any meaningful solution can only be achieved when a divergence-free velocity field is recovered, i.e. $\partial p/\partial t=0$. For a steady-state calculation this is not a problem. For unsteady flow problems, to achieve a fully time accurate solution, a divergence-free velocity must be attained at every physical time step. This can be achieved by using a dual-time stepping technique, subiterating the equations in a pseudotime domain to achieve a steady state solution at each physical time step.

In the present study, a cell centred finite volume approach has been adopted to discretize the governing equations. For each control volume $(i,\,j,\,k)$, equation (1) can be written as

$$\frac{\partial\boldsymbol{Q}_{i,j,k}A_{i,j,k}}{\partial t}=-\sum_{n=1,m(i,j,k)}\boldsymbol{F}_n\Delta l_n+\boldsymbol{B}A_{i,j,k}=-R\left(\boldsymbol{Q}_{i,j,k}\right) \tag{3}$$

where $\boldsymbol{Q}_{i,j,k}$ is the average value of $\boldsymbol{Q}$ in cell $(i,\ j,\ k)$ (stored at the cell centre), $A_{i,j,k}$ is the area of the cell, $\boldsymbol{F}_n$ is the numerical flux across its face n with a length of l_n.
To evaluate the inviscid numerical fluxes, Roe's flux function is adopted locally at each cell edge, assuming a one-dimensional Riemann problem in the direction normal to the cell edge, as follows:

$$\boldsymbol{F}_n = \tfrac{1}{2}\left[\boldsymbol{F}\left(\boldsymbol{Q}_n^+\right) + \boldsymbol{F}\left(\boldsymbol{Q}_n^-\right) - \boldsymbol{R}\left|\Lambda\right|\boldsymbol{L}(\boldsymbol{Q}_n^+ - \boldsymbol{Q}_n^-)\right] \tag{4}$$

where $\boldsymbol{Q}_n^+$ and $\boldsymbol{Q}_n^-$ are the reconstructed right and left states at edge n of cell $(i,\ j,\ k)$ and the quantities $\boldsymbol{R}$, $\boldsymbol{L}$ and Λ are the right, left eigenvectors and eigenvalues of the flux Jacobian $\boldsymbol{A} = \partial\boldsymbol{F}/\partial\boldsymbol{Q}$ evaluated by Roe's average state.
By discretizing equation (2) in time and omitting the subscripts for cell index, the first-order Euler implicit difference scheme, for example, can be employed:

$$\tfrac{(\boldsymbol{Q}A)^{n+1} - (\boldsymbol{Q}A)^n}{\Delta t} = -R(\boldsymbol{Q}^{n+1}) \tag{5}$$

To achieve a time-accurate solution at each physical time step in unsteady flow problems, equation (5) must be further modified to obtain a divergence free velocity field. This is accomplished by introducing a pseudotime derivative into the system of equations, as

$$\tfrac{(\boldsymbol{Q}A)^{n+1,m+1} - (\boldsymbol{Q}A)^{n+1,m}}{\Delta\tau} + I_{ta}\tfrac{(\boldsymbol{Q}A)^{n+1,m+1} - (\boldsymbol{Q}A)^n}{\Delta t} = -R(\boldsymbol{Q}^{n+1,m+1}) \tag{6}$$

where $\tau$is the pseudotime and $I_{ta} = \text{diag}[1\ ,\ 1,\ 1,\ 1,\ 0]$. The right side of the equation (6) can be linearized using Newton's method at the m + 1 pseudotime level to yield

$$\left[I_m A + \tfrac{\partial R(\boldsymbol{Q}^{n+1,m})}{\partial \boldsymbol{Q}}\right]\left(\boldsymbol{Q}^{n+1,m+1} - \boldsymbol{Q}^{n+1,m}\right) = -\left[I_{ta}\tfrac{\left(\boldsymbol{Q}^{n+1,m} - \boldsymbol{Q}^{n,m}\right)A}{\Delta t} + R\left(\boldsymbol{Q}^{n+1,m}\right)\right] \tag{7}$$

where $I_m = \text{diag}\left[\tfrac{1}{\Delta\tau} + \tfrac{1}{\Delta t}, \tfrac{1}{\Delta\tau} + \tfrac{1}{\Delta t}, \tfrac{1}{\Delta\tau} + \tfrac{1}{\Delta t}, \tfrac{1}{\Delta\tau} + \tfrac{1}{\Delta t}, \tfrac{1}{\Delta\tau}\right]$. When $\Delta(\boldsymbol{Q}^{n+1})^m = \boldsymbol{Q}^{n+1,m+1} - \boldsymbol{Q}^{n+1,m}$ is iterated to zero, the density and momentum equations are satisfied, and the divergence of the velocity at time level n + 1 is zero. The system of equations can be solved at each pseudotime step using an approximate LU factorization (ALU) scheme [7,8,10].

RESULTS AND DISCUSSION

To validate the numerical method, a simple experimental test case [12] involving a collapsing water column and its interaction with a small 3D box has been chosen and simulated. The water column with a height 0.55 m and a width of 1.22 m was initially confined to the left in a container of size 3.22 m × 1.0 m × 1.0 m (as shown in Figure 1) which was discretized with 161 × 50 × 50 cells. The confinement was then suddenly withdrawn at time $t = 0.0$s. Under the effect of gravity, the collapsing water front reached the solid box at around $t = 0.4$s and as it hits the wall of the box, a thin jet of

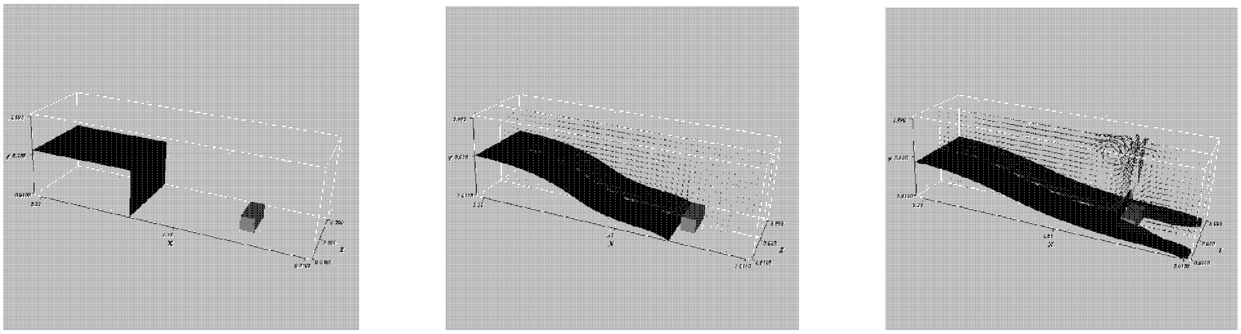

Figure 1: Simulation of water from a dam break hitting a fixed obstacle in an enclosed box at various times

water has formed at the front face of the box. At the same time, two streams of flow passed smoothly at both sides of the box and after some time, the water jet eventually collapsed onto the top of the box. During the simulation two water height gauges have been used: one in the water column and the other just in front of the small box. In Figure 2, the time history of water height at the two locations are compared with the experimental data for a total of six seconds. For the water height in the reservoir, the numerical results agree with the experimental data very well until water front flows back to the reservoir and for the water height probe just in front of the box, the numerical code has predicted the arrival of the water front accurately and the global trend also agrees well with the test data. As the viscosity has been ignored in the current simulation, the peak value of the water height in the second probe has been over-predicted. These differences between the numerical prediction and the experimental data will be further investigated by including viscous effects and also a grid refinement test.

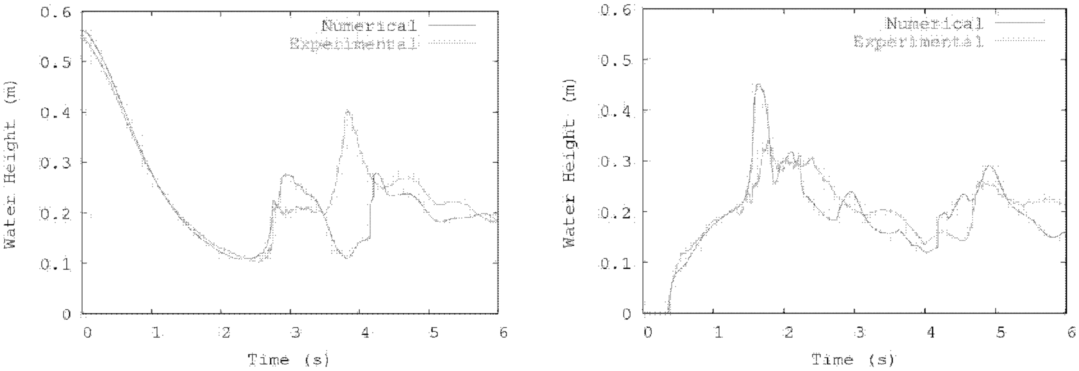

Figure 2: Vertical water heights in the reservoir (left) and the tank (right)

Future work will include the incorporation of a 3D Cartesian cut cell library for modelling free surface flows with solid stationary or moving bodies of complex geometry. Other results relevant to real coastal engineering flow problems will also be presented in the conference.

REFERENCES

1. Glimm J, Grove J, Lingquist B, McBryan O A, Tryggvason G. The bifurcation of tracked scalar waves. *SIAM J Sci Stat Comput*, 1988; **9(1)**: 61-79
2. Tryggvason G, Bunner B, Esmaeeli A. A front-tracking method for the computation of multiphase flows. *J Comp Phys*, 2001; **169**: 708-759
3. Hirt C W, Nichols B D. Volume of fluid (VOF) method for dynamics of free boundaries. *J of Comp Phys*, 1981; **39**: 201-225
4. Osher S, Sethian J A. Fronts propagating with curvature dependent speed: algorithms based on hamilton-jacobi formulations. *J of Comp Phys*, 1988; **79**: 12-49
5. Lafaurie B, Nardone C, Scardovelli R, Zaleski S, Zanet G. Modelling merging and fragmentation in multiphase flows with SURFER. *J of Comp Phys*, 1994; **113**: 134-147
6. Ubbink O. Numerical prediction of two fluid systems with sharp interface: [Ph.D Thesis]. Imperia College. 1997
7. Kelecy F J, Pletcher R H. The development of a free surface capturing approach for multidimensional free surface flows in closed containers. *J of Comp Phys*, 1997; **138**: 39-980
8. Pan D, Chang C H. The capturing of free surface in incompressible multi-fluid flows. *International J of Numer Methods in Fluids*, 2000; **33**: 203-222
9. Qian L, Causon D M, Mingham C G, Ingram D M. A Cartesian cut cell two-fluid solver for hydraulic flow problems. *ASCE J of Hydr Eng*, 2003; **129(9)**: 688-696
10. Qian L, Causon D M, Mingham C G, Ingram D M. A free-surface capturing method for two fluid flows with moving bodies. In: Proceedings of the royal society: 2006; A **462** (2065): 21-42
11. Qian L, Mingham C G, Causon D M, et al. Numerical simulation of wave power devices using a two fluid free surface solver. *Modern Physics Letters B*, 2005; **19(28-29)**: 51-54
12. Kleefsman K M, Fekken G, Veldman A E P, et al. A volume-of-fluid based simulation method for wave impact problems. *J Comp Phys*, 2005; **206**: 363-393