Transactions on Ecology and the Environment vol 12, © 1996 WIT Press, www.witpress.com, ISSN 1743-3541

A full numerical simulation of wave dynamics at coastal structures with the volume of fluid method

Z.A. Sabeur,[a] R.G. Beale,[b] V. Bovolin[c]
[a]*BMT Marine Information Systems Limited, Southampton SO14 3TJ, UK*
[b]*Oxford Brookes University, SCES, Oxford OX3 0BP, UK*
[c]*University of Salerno, Department of Civil Engineering, Fisciano, Italy*

Abstract

The volume of fluid method (VOF) of Hirt and Nichols[1] is applied in the simulation of transient waves at coastal structures where the fluid free surface gets mostly distorted at impact zone and important wave interactions with external boundaries take place[2]. The numerical stability of a VOF based code is maintained during the simulation of five wave periods which are generated by a weakly reflective boundary condition (WRIB). In this paper, the VOF method is not fully introduced but numerical techniques for solving the Poisson equations at arbitrary boundaries are discussed with cases of wave flows in a 35. m long and 1.5m deep pool with a slope are shown.

1 Introduction

Early 'Dam-break' flow simulations[3,4] show that the VOF technique is useful for the modelling of waves which freely interact with external boundaries and without the occurrence of numerical instability at the crucial moment of impact with structures[5]. This main advantage of the VOF method leads to the full numerical prediction of the dynamic interaction of the wave front with external boundaries. Our interest in such numerical exercise is in building a numerical algorithm which remains stable for a sufficiently long period of time, say, five wave periods, and successfully converges to predicting the type of wave interaction (or indeed, interactions) that occur at coastal zone. Such predictions could provide more understanding in the physics of waves with large vertical accelerations at non-porous (or porous) media and lead to more efficient design of sea defences and walls in hydraulic engineering.

A two-dimensional 'Dam-break' flow model developed by Sabeur *et al* [6] predicts the hydrodynamic pressure time histories and jet velocities on a vertical wall in a sloped rectangular tank. A transient wave caused by the release of a standing column of water hits the bottom of the wall and large vertical accelerations of fluid along the wall are computed. The simulation of the flow is performed until the fluid reaches hydrostatic equilibrium in the tank, and magnitude of velocities decrease towards vanishing values. This suggests that the VOF technique has a good potential for the simulation of waves at coastal structures and also, the study of confined flows by arbitrary boundaries with large eddies, recirculation arising from highly distorted interface and turbulence.

The model solves a system of Poisson equations for the dynamic pressures which are derived from the Navier-Stokes (NS) and mass conservation (MC) equations depending on the geometry and type of boundaries in the flow domain. The NS and MC equations are expressed respectively as follows:

$$\frac{\partial u}{\partial t} + u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} = -\frac{\partial p}{\partial x} + \gamma . \nabla^2 u + g_x \tag{1}$$

$$\frac{\partial v}{\partial t} + u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} = -\frac{\partial p}{\partial y} + \gamma . \nabla^2 v + g_y \tag{2}$$

and

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} = 0 \tag{3}$$

u and v are the velocity variables in the horizontal and vertical directions respectively, γ the kinematic viscosity and $g_{x,y}$ the horizontal and vertical components of the acceleration due to gravity. p is the reduced pressure which is related to the density of the fluid ρ, and the dynamic pressure P:

$$p = \frac{P}{\rho_{fluid}} \tag{4}$$

The Poisson equations can be derived from the combination of equations (1), (2) and (3) which are discretised in space and time by means of finite difference schemes. Many possible schemes can be used depending on the desired numerical stability and/or accuracy and the authors recommend Sharif and Bousnaina [7] for more details.

Transactions on Ecology and the Environment vol 12, © 1996 WIT Press, www.witpress.com, ISSN 1743-3541

In addition to the above, each Poisson equation should be derived in a way that it satisfies all possible type of external boundary encountered by the fluid during the flow simulation .

2 The Poisson equations

The model has been extended to dealing with several type of boundaries which are necessary for the simulation of realistic wave flows that can be validated with experimental data. For instance, the WRIB condition which generates a progressive wave with a given wave length λ and height *H* has been recently implemented by the inclusion of the Poisson equations at the inflow boundary cells, and the assumption of a horizontal free surface.

Firstly, the general discretised Poisson equation inside the fluid is simply derived as:

$$\frac{p_{i+1,j}^{n+1}-2p_{i,j}^{n+1}+p_{i-1,j}^{n+1}}{\delta x^{2}}+\frac{p_{i,j+1}^{n+1}-2p_{i,j}^{n+1}+p_{i,j-1}^{n+1}}{\delta y^{2}}=\frac{1}{\delta t}\left(\frac{A_{i,j}^{n}-A_{i-1,j}^{n}}{\delta x}+\frac{B_{i,j}^{n}-B_{i,j-1}^{n}}{\delta y}\right) \quad (5)$$

p is the discretised reduced pressure at time level *n+1* and is represented at the centre of each computational cell. *A* and *B* are the discretised non linear finite difference terms of the NS equations at time time level *n*, δx and δy the horizontal and vertical grid spacing respectively and δt the time step. In this case, a rectangular uniform computational grid is assumed for simplification, however similar equations to equation (6) can be derived for a non uniform grid without major difficulty.

The Poisson equations at the boundaries are similar to equation (5), nevertheless special physical processes and considerations such as friction and porosity must be taken into account for each type of boundary. We believe that boundary cells are classified into the following five categories for the case of incompressible flow:

- non porous
- Porous
- Free surface
- Inflow
- Outflow

The Poisson equations at non porous boundary cells can be obtained by considering rigid free-slip or no-slip flow conditions. For example, in figure 1 which represents the case of a flow at a non-porous free-slip corner boundary, the velocities *v'* and *u'* in the virtual cells (i+1,j) and (i,j-1) must be respectively equal to the velocities *u* and *v* in the real cell (i,j).

Transactions on Ecology and the Environment vol 12, © 1996 WIT Press, www.witpress.com, ISSN 1743-3541

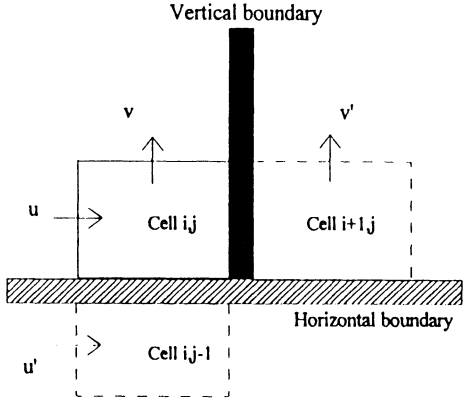

Figure 1: Free-slip conditions at a non-porous boundary

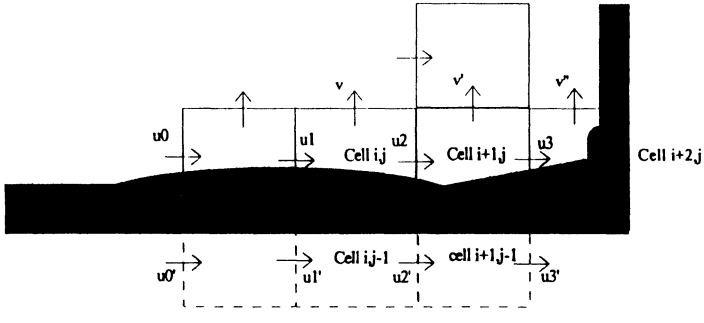

Figure 2: Free-slip conditions at arbitrary shaped non-porous boundary

Transactions on Ecology and the Environment vol 12, © 1996 WIT Press, www.witpress.com, ISSN 1743-3541

In addition to that, the velocities at both horizontal and vertical walls must be set to zero in order to satisfy impermeability. Hence, the corresponding Poisson equation at cell (i,j) becomes:

$$\frac{-p_{i,j}^{n+1}+p_{i-1,j}^{n+1}}{\delta x^{2}}+\frac{p_{i,j+1}^{n+1}-p_{i,j}^{n+1}}{\delta y^{2}}=\frac{1}{\delta t}\left(\frac{-A_{i-1,j}^{n}}{\delta x}+\frac{B_{i,j}^{n}}{\delta y}\right) \tag{6}$$

In the case of rigid no-slip conditions, the virtual velocities are simply set to:

$$u'=-u \text{ and } v'=-v \tag{7}$$

Furthermore, similar principles can be applied to both real and virtual velocities in the case of arbitrary shaped boundaries[8]. In figure 2 and in particular at cell (i+1,j), the velocities *u2*, *v'* and *u3* are used to derive the Poisson equation. Had the sloped part of the boundary covered more than half of the right hand face of cell (i+1,j), the velocity *u3* would not have been used in the calculation of the pressures. In cell (i+2,j) for instance, only *v''* and *u3* are needed to derive the Poisson equation. However, the virtual velocities must be used whenever the boundary edge clearly crosses a cell from the left hand side face to the right hand side. (or from the bottom side to the top side)
At cell (i+1,j), the Poisson equation is:

$$\frac{p_{i+2,j}^{n+1}-2p_{i+1,j}^{n+1}+p_{i,j}^{n+1}}{\delta x^{2}}+\frac{p_{i+1,j+1}^{n+1}-p_{i+1,j}^{n+1}}{\delta y^{2}}=\frac{1}{\delta t}\left(\frac{A_{i+1,j}^{n}-A_{i,j}^{n}}{\delta x}+\frac{B_{i+1,j}^{n}}{\delta y}\right) \tag{8}$$

At other types of boundaries such as the ones classified earlier, the Poisson equations are derived from specific conditions for the pressures, the velocities or both. At the interface of fluid and air for example, the pressure is maintained to a desired fixed value; thus a simple linear interpolation (or extrapolation) can be used to represent a free surface cell pressure equation. Also, the continuity of shear stress must be enforced by setting the tangential and normal velocities at all mixed air/fluid cells.

The location of the interface in time is tracked by using the so-called *F* function which computes the fractional volume of fluid in each computational cell. *F* varies between zero and one and represents a full cell with value one, an empty cell with value zero and a free surface cell or an air bubble with intermediate values. The orientation of the interface is uniquely determined and the free surface boundary conditions can be implemented, which clearly is the main advantage of the VOF method and the key to modelling highly distorted waves at external boundaries. Also, wall adhesion, surface tensions and contact angle features can be added in the model for the study of supercritical flow at structures.

2.1 The WRIB condition and the progressive wave

The implementation of a wave maker to generate a progressive wave of period T and height H can be efficiently simulated by the WRIB condition settings.

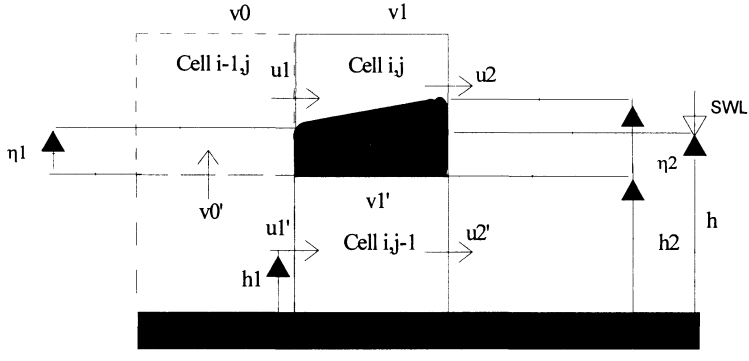

Figure 3: WRIB conditions at the wavemaker

In figure 3, cell (i,j-1) is a typical inflow cell, where $u1'$, $v1'$ are imposed velocities by the wave maker of a progressive wave. They are chosen to have the following behaviour in space and in time:

$$u1' = \frac{H}{2}\omega\frac{\cosh(kh_1)}{\sinh(kh)}\sin(\omega t) \tag{9}$$

and

$$v1' = \frac{H}{2}\omega\frac{\sinh(kh_2)}{\sinh(kh)}\cos(\omega t + \frac{k\delta x}{2}) \tag{10}$$

where k is the wave number, δx the grid cell spacing in the horizontal direction and h the still water depth.

The free surface at cell (i,j) is assumed to be horizontal and water elevations η_1 and η_2 are simply imposed by :

$$\eta_1 = H/2\ \sin(\omega t)\ \ \&\ \ \eta_2 = H/2\ \sin(k\delta x - \omega t) \tag{11}$$

The wave number is computed from the wave dispersion relation theory:

$$\omega = \sqrt{gk\tanh kh} \tag{12}$$

Also, the wave propagation into each inflow cell must satisfy flow continuity, then the reflected wave velocities and elevations respectively satisfy the following criteria:

$$\frac{\partial \bar{u}}{\partial t} - C\frac{\partial \bar{u}}{\partial x} = \frac{\partial \bar{u}_{\inf low}}{\partial t} - C\frac{\partial \bar{u}_{\inf low}}{\partial x} \tag{13}$$

and

$$\frac{\partial \eta}{\partial t} - C\frac{\partial \eta}{\partial x} = \frac{\partial \eta_{\inf low}}{\partial t} - C\frac{\partial \eta_{\inf low}}{\partial x} \tag{14}$$

C is the wave celerity assumed for both incoming and reflected waves.

Consequently, the Poisson equation relative to the inflow cell type can be derived by the discretisation of equations (13) and (14) in order to compute the non linear terms A and B which appear in equation (5).

$$\frac{p_{i+1,j-1}^{n+1} - p_{i,j}^{n+1}}{\delta x^{2}} - \frac{p_{i,j-1}^{n+1} - p_{i,j-2}^{n+1}}{\delta y^{2}} = \frac{1}{\delta t}\left(\frac{A_2^{n'} - u_2^{n'}}{\delta x} + \frac{v_1^{n'} - B_1^{n'}}{\delta y}\right) \tag{15}$$

The Poisson equations for an outflow boundary can be derived in a similar way to the inflow case, providing that the right hand side terms of both equations (13) and (14) are switched off. If, however, the wave propagates into a porous medium, different NS equations can be derived with adequate flow transport and friction criteria using added flow mass coefficients and filter-discharge velocities.

3 Numerical applications

The resulting Poisson equations for the flow are solved by the Gauss-Seidel technique which requires 1000 iterations in order to converge to the solution for the pressures at each time step. 150x90 uniform rectangular grid cells are used in the computation, and the time step is of the order of 0.001 sec for stability. 2mins CPU time per time step, in a four processor sparc workstation, is required in the computation. This suggests that the computationally intensive task for solving such equations could be reduced substantially if one refers to the use of parallel machines with adapted algorithm solvers[9].

The simulation of a progressive wave with period $T=2.8\ sec$ and height $H=1.m$ has been achieved by the WRIB condition. The numerical model is numerically stable for more than five wave propagations in a 35.0 m long and 1.5m deep pool with a 1:4 slope boundary. However, numerical diffusion and instability at the inflow region occur when strongly reflected waves interact with the incoming wave and flow continuity violated.

Transactions on Ecology and the Environment vol 12, © 1996 WIT Press, www.witpress.com, ISSN 1743-3541

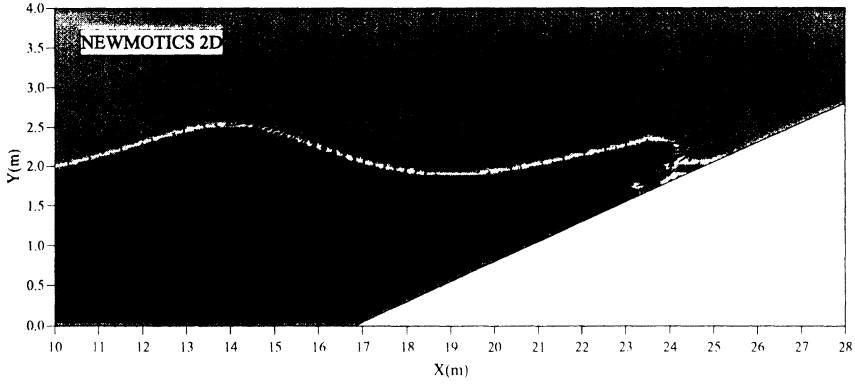

Figure 4: Breaking wave at a sloped boundary, t = 12.10 sec

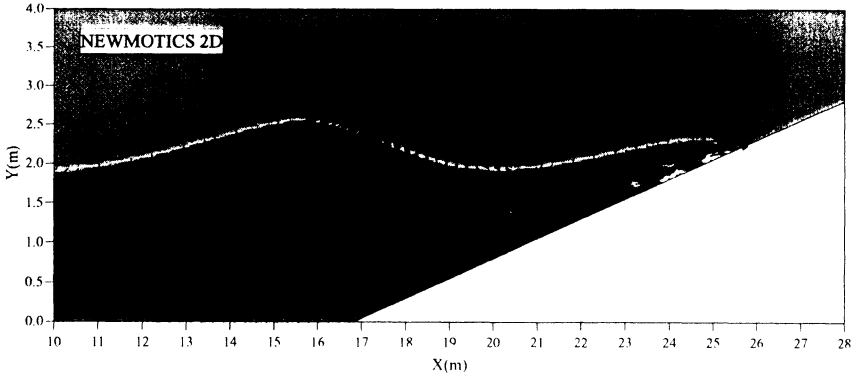

Figure 5: Collapsing wave at a sloped boundary, t = 12.40 sec

Transactions on Ecology and the Environment vol 12, © 1996 WIT Press, www.witpress.com, ISSN 1743-3541

In figure 4, at t=12.10 sec, the breaking of the wave front is clearly seen by the presence of velocity jets and overlaping boundary layers with trapped air bubbles. The wave collapses back on to the slope due to energy dissipation and gravity 0.30 sec later, as shown in figure 5.

4 Conclusion

The mathematical foundations for modelling transient waves at arbitrary shaped structures has been discussed and adapted to the powerful VOF technique which simulates complex flows in confined boundaries. The stability of the method enables the numerical measurement of physical observables that are responsible for wave interactions with external boundaries. This research is currently in progress and more numerical results which are to be compared with experiment will reveal the extent of the use of the VOF method.

Acknowledgements

This work has been been partly achieved at the Edinburgh parallel Computing Centre where preliminary wave simulations using the CM200 machine have been completed. Z.A.Sabeur would like to thank, BMT Marine Information Systems Ltd for generously providing Internet link to Edinburgh and Oxford in order to run the remaining wave test cases.

References

1. Hirt, C.W. & Nichols, B.D. Volume of fluid method for the dynamics of free boundaries, *Journal of Computational Physics*, 1981, **39/1**, 201-225.

2. Peregrine, D.H. Pressure on breakwaters: A forward look. *University of Bristol Mathematics research report,* **AM-93-17**, 1994.

3. Nichols, B.D., Hirt, C.W. & Hotchkiss, R.S. SOLA-VOF: A solution algorithm for transient fluid flow with multiple free boundaries. *Los Alamos National Laboratory report,* **LA-8355**, 1980.

4. van der Meer, J.W., Petit, H.A.H., van den Bosch, P., Klopman, G. & Broekens, R.D. Numerical simulation on and in coastal structures, in ICEE/ 92, pp. 1448-1460, Venice, 1992.

5. Ramshaw, J.D. & Trapp, J.A. A numerical technique for low speed homgenous two phase flow with sharp interfaces. *Journal of Computational Physics,* **21**, 21-43, 1976.

6. Sabeur, Z.A., Roberts, W. & Cooper, A.J. Development and use of an advanced numerical model using the volume of fluid method for the design of coastal structures, in Numerical methods for fluid dynamics V (ed K.W.Morton & M.J.Baines), pp. 565-573, Oxford Science Publications, 1995.

7. Sharif, M.A.R. & Busnaina, A. *Journal of Computational Physics,* **74**, 143-176, 1988.

8. Viecelli, J.A. A method for including arbitrary external boundaries in the MAC incompressible fluid computing technique, *Journal of Computational Physics,* **4**, pp. 543-551, 1969.

9 Sabeur, Z.A. *A parallel computation of the Navier-Stokes equations for the simulation of free surface flows with the volume of fluid method*, in Applied parallel computing, (Ed J.Dongara & J.Wasniewski), Lecture Notes in Computer Science No 1041, Springer-Verlag, pp. 483-492, Denmark, 1996.