CHAPTER 31

WAVE DYNAMICS AT COASTAL STRUCTURES: DEVELOPMENT OF A NUMERICAL MODEL FOR FREE SURFACE FLOW

Zoheir A. Sabeur[1], N William H.Allsop[2], Robert G.Beale[3], John M.Dennis[4]

Abstract

The development of third generation wave models is needed for a detailed study of wave dynamics and impact at coastal structures. This would require the modelling of wave flows with high distortion of the free surface at confined boundaries. In our opinion, the Volume of fluid method, which uses concepts of local advection of fluid in free surface flow modelling, is the prime candidate for simulating realistic flows at sea defences and walls. In this paper, the numerical techniques for the simulation of waves with highly distorted water/air interfaces at a slope, using the volume of fluid method, are considered

Introduction

The volume of fluid method (VOF), originally developed by Hirt and Nichols [1], can be used for the study of transient waves within confined structures. The possibility for simulating the interaction (or interactions) of waves at structures with complex geometry leads to a more realistic modelling of wave impact at breakwaters and coastal defences. Furthermore, a stable numerical simulation of such wave dynamics can provide important information on impact pressures, jets and wave flows at specified locations of the coastal structure.

1) Principal Modeller, BMT Marine Information Systems Ltd, Grove House, Meridians Cross, 7 Ocean Way, Ocean Village, Southampton, SO14 3TJ, UK.
2) Coastal Structures Manager, HR Wallingford Ltd, Oxfordshire OX10 8BA, UK.
3) Senior Lecturer, Oxford Brookes University, Civil Engineering Division, Oxford, OX3 0BP, UK.
4) Principal Lecturer, Oxford Brookes University, Civil Engineering Division, Oxford, OX3 0BP, UK.

Thus, the numerical approach based on the VOF technique for the simulation of transient waves can lead to a good design for the construction of sea defences that sustain wave forces encountered during severe ocean, sea and atmospheric conditions.

In previous years, two dimensional depth integrated models have been widely used in the study of wave action at vertical walls and slopes [2]. However they are limited by the fact that the free surface is assumed to remain a single valued function of space in all flow cases. Vertical fluid accelerations which are large in magnitude and short in duration, during wave impact, cannot be modelled and the free surface is implicitly maintained as a simply connected function of space. These first generation type of models are well established in coastal engineering. They can provide solutions to problems involving wave propagation at structures with simple geometry and in flow cases where breaking and overturning waves do not take place. They are also important for the study of solitary waves and can be used in the development of the next generation wave models. [3], [4]

The speed and storage memory of modern computers have allowed scientists to develop the second and third generation wave models, and despite the task for a large amount of numerical operations, driven by new mathematical algorithms, the total CPU time used by such computers can be optimised and lead to an efficient study of wave dynamics at coastal structures.

Second generation wave models based on Boundary Integral Methods (BIM) which were developed by Peregrine *et al* , and others [5], raised important questions on the physics of impact pressures at vertical walls. Peregrine showed the existence of large magnitude vertical accelerations of a wave front at impact and the possible mechanisms by which air bubble entrapment could be governed. The existence of short timed (~ 1msec) oscillatory impact pressure peaks at specific locations of the vertical wall suggests the existence of several types of wave interactions with the wall. Therefore, a detailed numerical investigation needs to be carried out.

As a result, the third generation wave models which describe the dynamics of waves before, at and after impact have been developed, and recent simulations of breaking and overturning waves at vertical walls and slopes have been completed.[6].

Mathematical formulation

The general equations for fluid motion under transient conditions are given by the Navier-Stokes equations (NS). They incorporate fluid mass and momentum conservation and take into account of the external forces that are applied on the fluid body.

In two dimensional cartesian coordinates, the NS equations are given by the following differential equations:

Conservation of mass:

$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} = 0 \tag{1}$$

Conservation of momentum:

$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} = \rho F_x + \sum_{r=x,y} \frac{\partial \sigma_{xr}}{\partial r} \tag{2}$$

and

$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} = \rho F_y + \sum_{r=x,y} \frac{\partial \sigma_{yr}}{\partial r} \tag{3}$$

F_x and F_y represent the components of the resultant external force applied on the fluid in the x (horizontal) and y (vertical) directions respectively. σ is the stress tensor and ρ is the density of the fluid. u and v are the velocity field components in the x and y directions respectively. t is the time variable.

In the case of a Newtonian incompressible fluid the density ρ is constant in time and the stress tensor σ is assumed to vary linearly with the fluid deformation rate.

Thus:

$$\frac{1}{\rho}\begin{bmatrix}\sigma_{xx} & \sigma_{xy} \\ \sigma_{yx} & \sigma_{yy}\end{bmatrix} = \begin{bmatrix}-p & 0 \\ 0 & -p\end{bmatrix} + \gamma\begin{bmatrix}2\frac{\partial u}{\partial x} & \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \\ \frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} & 2\frac{\partial v}{\partial y}\end{bmatrix} \tag{4}$$

where p is the reduced pressure and γ is the kinematic viscosity. (All variables are used in (SI) units.). As a result, equations (1), (2) and (3) simplify and can be written in a compact vectorial form:

$$divU = 0 \tag{5}$$

and

$$\frac{\partial U}{\partial t} + (U \cdot grad)U = F - grad(p) + \gamma \nabla^2 U \tag{6}$$

U and F are the velocity and external force vector fields respectively.

Numerical Model

Sabeur *et al* developed a new VOF based model in recent years [7]; this model discretises the NS equations in both time and space by means of finite difference techniques. Equations (5) and (6) can be written as follows:

$$divU^{n+1} = 0 \tag{7}$$

and

$$\frac{U^{n+1} - U^n}{\delta t} + grad(p^{n+1}) = Q^n \tag{8}$$

δt is the time step and Q^n involves the finite difference advective terms and the external force F of the NS equations:

$$Q = -(U \cdot grad)U + \gamma \nabla^2 U + F \tag{9}$$

Various finite difference schemes can be implemented in the spatial discretisation of Q. As far as numerical stability and accuracy are concerned, each of the schemes have their advantages and drawbacks. The choice of the finite difference scheme greatly depends on the computing task required for the wave modelling case. ie, Number of wave propagation periods, wave length, boundary geometrical complexity.

When combined, equations (7) and (8) lead to the Poisson equation (PE) for the pressures:

$$\nabla^2 p^{n+1} = div(\frac{U^n}{\delta t} + Q^n) \tag{10}$$

The volume of fluid method

The study of wave motion with a highly distorted free surface needs the implementation of the VOF method which tracks the fluid locally in space. For a specified rectangular grid and time step, fluid fluxes are computed at each cell face. The fractional amount of fluid per cell is then determined by the net flux of fluid advected in both vertical and horizontal directions.

The flux calculation leads to the update of the F function in time. That is the fractional volume of fluid at each cell centre point. Thus, a zero value of F in a cell means that it is empty while a unit value of F corresponds to a cell full of fluid. An intermediate value of F between zero and one normally represents a free surface cell, or indeed a trapped bubble. In strict numerical modelling terms, however, a true free surface cell is a cell that has at least one empty neighbouring cell. In figure 1 for example, the cell where $F= 0.99$ is not considered as a free surface cell because its four nearest neighbour cells are not empty. Therefore the PE is applied for such type of cell as if it was a full cell.

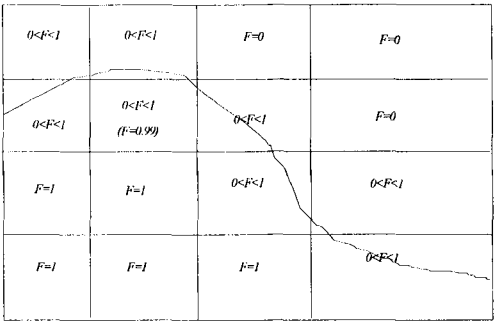

Figure 1. Illustration on the type of fluid and void cells considered by the VOF wave model.

In addition to the above, a numerical correction is performed on the F function values at the end of each computational cycle because, usually, a unit value of the F function (or, indeed a zero value) cannot be reached accurately by computational means. Instead, it is rounded to one (or zero) if it reaches a value of 1+/-ε. (or +/-ε). In practice, and for an efficient modelling of the fluid interface, ε should be no bigger than 0.00001.

Boundary conditions

In the VOF method, pressure and velocity equations must be specified.

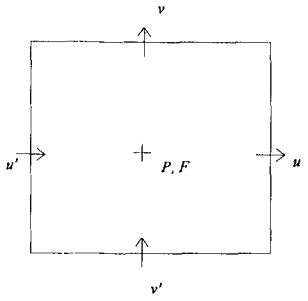

Figure 2. A typical 2D VOF cell with six dynamic variables

Figure 2 shows the location of the six variables used in a staggered finite difference grid and computed for each time level n. In the case of a full cell located inside the flow (meaning, that the cell is totally surrounded by full cells), equations (8) and (10) are used to compute the velocities and pressures at time level $n+1$ respectively. Then, the F function value in the cell can be updated and the fluid advected in time. However, in the case of boundary cells, the computation of the pressures, the velocities and function F becomes tedious and time consuming.

Boundary cells can be classified as follows:

- Ordinary free surface cell
- Non-ordinary free surface cell
- Inflow cell
- Full cell at a vertical/horizontal/sloped rigid boundary

For each boundary cell, apart from the free surface cells, pseudo Poisson equations can be implemented by setting fluid mass conservation and virtual velocities inside local boundaries [8]. For example, rigid free-slip (or no slip) boundary conditions can be established for solid boundaries. The normal velocities to the boundaries and the gradient of the tangential velocities are set to zero. Parallel (or anti-parallel) and equal in magnitude virtual velocities are set for rigid free-slip (or no-slip) boundary conditions.

Free surface cells are provided with different pressure equations which involve the orientation and curvature of the interface, the nearest pressure inside the fluid, the local atmospheric pressure at the free surface and surface tension. The continuity of normal and tangential shear stresses is also enforced. A typical pressure equation at the free surface is given by:

$$-p + 2\gamma \frac{\partial U_n}{\partial r_n} = -p_{Air} \frac{\rho_{Air}}{\rho} + \frac{2\varphi\psi}{\rho} \tag{11}$$

supplemented with a condition on stresses at the interface:

$$\gamma\left(\frac{\partial U_n}{\partial r_t} + \frac{\partial U_t}{\partial r_n}\right) = 0 \tag{12}$$

φ and ψ are the free surface tension and curvature respectively. p_{Air} and ρ_{Air} are the reduced pressure and density of air respectively. U_n and U_t are the normal and tangential velocities to the free surface. The reduced pressure at the interface, p, is interpolated (or extrapolated) from the nearest dynamic pressure inside the fluid and normal to the interface. It can be located from the orientation of the free surface. Hence, the nearest interpolation dynamic pressure can be found either below or above the fluid interface, or can be located to the left or to the right of the interface. The orientation and curvature of the interface are computed by the F function local gradients corresponding to nearest neighbouring cells to the free surface.

This approach for setting the boundary conditions at the free surface is used by most VOF methods, however the boundary conditions for cells which intercept the slope are not straightforward.

In the new VOF wave model, the conditions on the orientation of the interface, given by the local gradients of F, are slightly modified at air/water/slope boundary cells. Virtual values of F, inside the sloped structures, are given *a priori* . Then, a stable air/water/slope interface can be modelled.

Further numerical development of air/water/slope boundary conditions is needed. Applications of flows with the VOF method can be then extended to more complicated type of interface. For instance, in cases of floating and moving objects in fluids.

The simulation of progressive waves at sloped structures can be generated by a weakly reflective inflow boundary condition (WRIB). [6], [8]. Continuity of flow is enforced at the boundary and the free surface is assumed horizontal at all time. The velocity vector field U and water elevation η at the inflow boundary must satisfy the following conditions:

$$\frac{\partial U}{\partial t} - C\frac{\partial U}{\partial x} = \frac{\partial U_{in}}{\partial t} - C\frac{\partial U_{in}}{\partial x} \tag{13}$$

and

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

C is the wave celerity. U_{in} and η_{in} are the inflow velocity field and water elevation (above or below still water level) respectively. For example, they take the following sinusoidal forms:

$$U_{in}(x,y,t) = \frac{H\omega}{2\sinh(kh)}\left[\cosh(ky)\sin(\omega t) + j\sinh(ky)\cos(\omega t + kx)\right] \tag{15}$$

and

$$\eta_{in}(x,t) = \frac{H}{2}\sin(kx - \omega t) \tag{16}$$

h is the still water level (SWL), ω the angular wave frequency, k the wave number and H the total wave amplitude. ($j = \sqrt{-1}$.)*

*(*The use of imaginary complex j in equation (15) should not be confused with grid cell coordinate j.*)

Equations (13) and (14) are discretised forward in time and solved one time step earlier than the pressure and velocity equations. In this manner, the velocity field U and elevation η, can be provided for the computation of term Q, which appears in equation (8) at time level n. Water elevation η enables the computation of F at the inflow boundary free surface cell. At the free surface, F is given by:

$$F = \eta + h - \sum_{j=1}^{j=N} F_{ij}\delta y_{ij} \tag{17}$$

N values of F_{ij} are found strictly equal to unity in a water column, j, which is adjacent to the inflow boundary column. δy_{ij} is the height of cell ij.

The combination of PE, the pseudo Poisson equations at the boundaries, and the free surface pressure equations leads to a system of M equations with M unknowns. M is the number of grid cells occupied by the fluid where F is greater than zero. Once the pressure equations are computed at a specified time level, the new velocity field is calculated, the F function is updated and the fluid advanced accordingly. The updated values of F provide the new value of M.

In an early simulation of a dam-break flow by the VOF method, the Gauss-Seidel (GS) iterative scheme was used to solve the pressure equations. The GS algorithm requires a computational time that is proportional to M^2. For M smaller than ~10000, the algorithm converged after approximately 1200 iterations.

Numerical simulations, results and discussion

The speed of the pressure solver can be improved by adapting successive over-relaxation techniques (SOR) to the GS algorithm. However, as M gets larger with high resolution grids, or when diagonal dominance of the PE deteriorates, the SOR algorithm slows down and does not successfully converge. Powerful methods such as the Conjugate gradient (CG) or Lanczos (LZ), should be used to overcome this problem. The pressure field should be therefore represented in vectorial form and the structure of the corresponding Poisson matrix updated at each time step. Preliminary attempts to the matrix formulation, using parallel processors, [9] have been carried out and adapted to the VOF wave model. The simulation of a Dam-break flow showed similar results to the early calculations with the GS method.

The simulation of progressive waves at a slope, using the WRIB condition, has been completed. The stability of the VOF model over several wave periods and for two cases of slopes, in a 4m deep and 35m long rectangular tank, has been achieved. The computation of the pressure equations was performed on a uniform 150x80 rectangular grid and the minimum time set to 0.001 sec. In this early simulation, surface tension was not included and the pressure at the interface was set to zero. (void representation). The wave characteristics for each cases of slopes are given in the table below:

Wave parameters	T (sec)	H(m)	SWL(m)	Slope	Surf ξ
1st case	2.8	1.	1.5	1:3	0.3
2nd case	2.8	1.	1.5	1:4	0.2

Table 1. Wave characteristics used for model tests.

According to Battjes theory on breakers[10], the first and second wave test cases correspond to flows with spilling breakers. Spilling breakers occur with a surf similarity coefficient ξ smaller than 0.5, which is the case in our study. Large magnitude impact pressures are not encountered in such flows but distortions of the free surface with large velocity jets at air/water/slope boundaries do take place. In the first wave test case, shown in figure 3 for example, air entrapment at the slope region occurs at *t=11.65 sec.* Maximum velocity jets reaching approximately 5 m/s at the wave crest are calculated. The wave then spills along the slope, as shown in figure 4, and the front of the spill changes direction due to gravity. This change in the direction of motion causes the top of the wave to break and overturn, as shown in figure 5 at *t=13.60 sec.* Additional air bubbles are then trapped inside the fluid and lead to the formation of velocity jets that are parallel to the slope plane and persist in time. (Figure 6.) The kinetic energy of the wave flow region driven by the jets, decreases gradually in time and causes the collapse of part of the wave crest back onto the slope, as shown in figure 7. This feature of the model clearly illustrates the potential modelling of transient waves by the VOF technique. Similar observations to the above equally apply to the second wave test case. They are shown in figures 8-12.

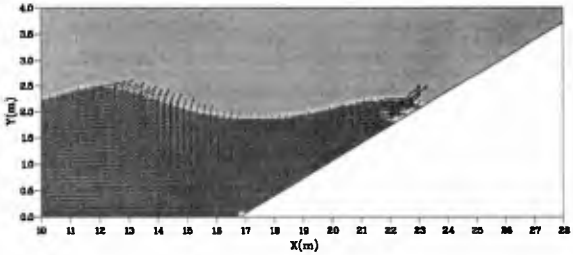

Figure 3. t=11.65 sec, slope=1:3

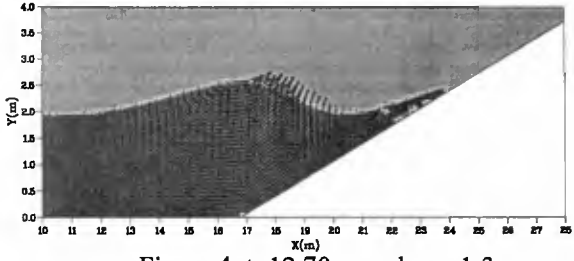

Figure 4. t=12.70 sec, slope=1:3

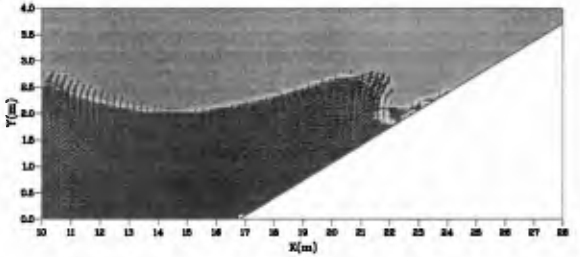

Figure 5. t=13.60 sec, slope=1:3

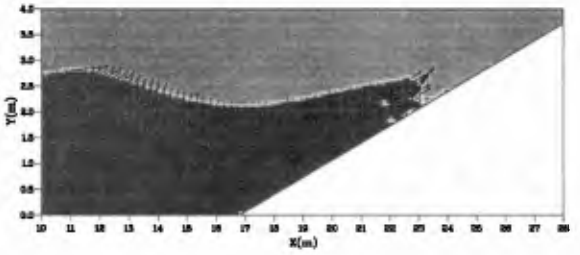

Figure 6. t=14.19 sec, slope=1:3

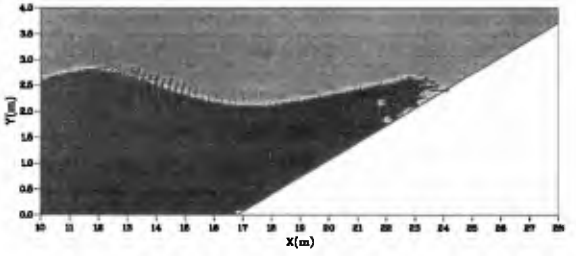

Figure 7. t=14.30 sec, slope=1:3

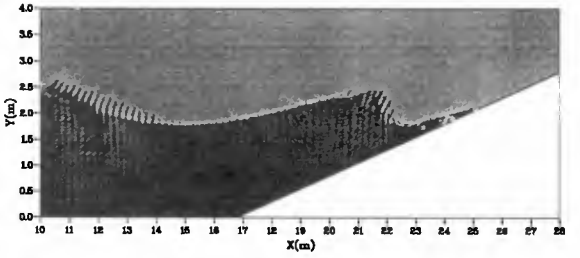

Figure 8. t=11.09 sec, slope=1:4

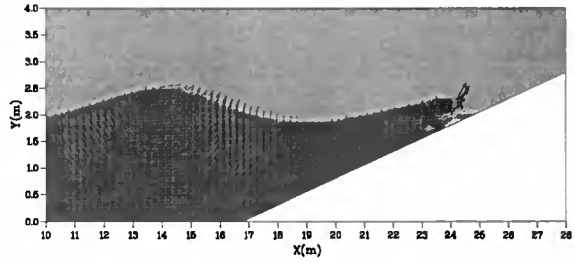

Figure 9. t=12.10 sec, slope=1:4

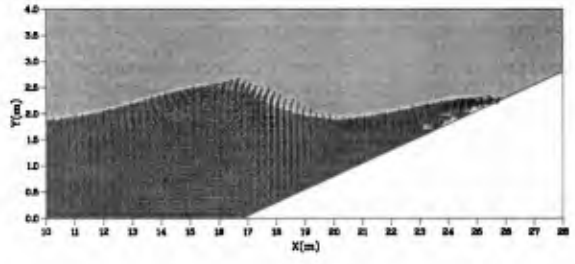

Figure 10. t=12.40 sec, slope=1:4

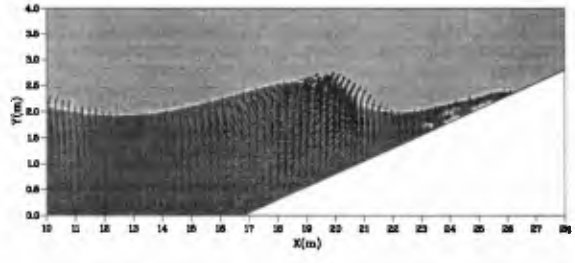

Figure 11. t=13.30 sec, slope=1:4

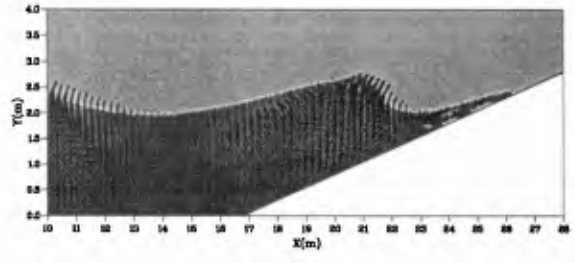

Figure 12. t=13.60 sec, slope=1:4

Conclusion and recommendations

This work clearly demonstrates the potential of the VOF implementation to the NS equation for the study of wave dynamics within confined domains, i.e. coastal structures. The newly developed model is capable of simulating the full process of wave interaction with structures. Wave impact generated from collapsing and plunging breakers can be equally modelled and dynamic pressures and jets predicted at specified locations of the structure. The model also provides more understanding in wave dynamics at coastal structures which in turn helps to promote improved design methods.

In order to realise the full potential of VOF based wave models in coastal engineering , the following research and development is recommended:

- Implementation of the Conjugate gradient or Lanczos algorithms in the pressure solver.
- Development of boundary conditions at water/porous medium interfaces.
- Implementation of the compressible NS equations for air entrapment dynamics study.
- Extension of the theory to 3D for a better understanding of wave interactions.
- Calibration and validation of numerical data with laboratory results.
- Comparison of numerical predictions of wave dynamics with field data.

Acknowledgements

This work was originally funded by Oxford Brookes University (OBU) and the UK Department of the Environment (Contract no. CI 39/5/86). The authors are grateful to A.J.Cooper and W.Roberts (HR Wallingford) for useful discussions during early development of the model. Z.A.Sabeur is thankful to A.E.P.Veldman of the University of Groningen, the Netherlands, for guidance on free surface modelling and impact dynamics at boundaries, L.Zieleznik (OBU), for processing the wave model data by AVS/Gsharp graphics, and the technical staff of BMT Marine Information Systems Ltd for support in the visualisation and transfer of .cgm files into Word 6. documents. The authors also thank A.Bracci-Laudiero of the University of Naples, Italy, for assistance in the preparation of the conference presentation, and V.Bovolin of the University of Salerno, Italy, for research collaboration. X.He (OBU) and M.Waller (HR Wallingford) are also thanked for the progress made in the implementation of the SOR technique to the pressure solver.

References

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

[2] Allsop, N.W.H *et al.* (1988). Development and application of a mathematical model on wave action on steep slopes. Proc. 21st ICCE/ 88, Malaga. publn. ASCE. New York

[3] McConnell, K.J *et al.* (1995). Wave dynamics at near port and coastal structures. Hydraulics Report IT422 HR Wallingford, UK.

[4] Sabeur, Z.A and Dennis, J.M. (1995). Wave Dynamics at Coastal Structures: Numerical Methods for the Simulation of flows with moving interfaces. Hydraulics Report IT428, HR Wallingford, UK .

[5] Cooker, M.J. and Peregrine, D.H. (1995). Pressure impulse theory for liquid impact problems. Journal of Fluid Mechanics, 297, 193-214.
(Also, Peregrine, D.H. (1994). Pressure on breakwaters: A forward look. University of Bristol Mathematics research report, AM-93-17, Bristol, UK.)

[6] van der Meer, *et al.* (1992). Numerical simulation on and in coastal structures, Proc. 23rd ICCE/ 92, 1448-1460, Venice. publn. ASCE, New York.

[7] Sabeur, Z.A., *et al.* (1995). 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), Oxford University Press, UK, 565-573.

[8] Sabeur, Z.A., *et al.* (1996). A full numerical simulation of wave dynamics at coastal structure with the VOF method. In, Hydraulic Software VI, Computational Mechanics Publications. (Ed W.R.Blain), Southampton, UK, 395-404.

[9] Sabeur, Z.A. (1995). 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, Denmark, 483-492.

[10] Battjes, J.A. (1974). Surf similarity, Proc. 14th ICCE/74., 466-480, Coppenhagen. publn. ASCE, New York.