土木学会第52回年次学術講演会（平成９年９月）

II －3

Numerical Simulation of Waves on a Submerged Reef

Graduate School of Science and Engineering, Saitama University Student Member Qun Zhao
Department of Civil Engineering, Saitama University Member K.Tanimoto
ditto Member Vu Than Ca
ditto Member H.Nakamura

Introduction

Submerged reef type breakwaters prevail the conventional coastal protection works owing to their remarkable functions of dissipating incoming wave energy, the capacity of water exchanges behind the breakwater and the preservation of the natural scenery.

However, the extremely complicate wave manner on a submerged reef caused by strong nonlinearity of waves and the wave breaking restricts the traditional study of wave transformation to small scale model tests. Thanks to the booming progress of computer science, the last twenty years have witnessed a remarkable growth of numerical modelling fueled by the advent of large, faster computers that are readily available and economical to use.

The well known numerical models which are commonly used to simulate strong nonlinear wave motions are the Nonlinear Shallow Water Equations (NSWE) and the Boussinesq models. Nevertheless, their assumption of depth uniform velocity makes them impossible to predict the sharp increase of velocities near the crest of a breaking wave which is the most important part of wave energy and momentum flux.

Abandon the above assumption, the present work is based on the Navier- Stokes equations and the Volume of Fluid (VOF) method which can give full field wave variables and potential to describe complicate water surface caused by wave breaking. According to the numerical calculations and the physical model tests, it was found that the model can give rather accurate results not only for water surfaces and velocity profiles, but also the correct breaker type and breaker position. It is expected that by further improvements, the numerical model could give more detail information of a breaking wave.

Governing Equations

The governing equations for viscous incompressible 2D-flow are the continuity equation:

$$\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{1}$$

and the Navier-Stokes equations:

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu(\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}) \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y} = -\frac{1}{\rho}\frac{\partial p}{\partial y}+\nu(\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2})+g \tag{3}$$

where ρ, ν are the density and the kinematic viscosity of the fluid respectively. The terms u, v, are the velocity components in horizontal(x) and vertical(y) directions. g is the gravitational acceleration, p is the pressure, and t is the time.

In volume of fluid method (VOF), a function F is used to define the fluid region. The physical meaning of F function is the fractional volume of the cell occupied by fluid. A unit value of F would correspond to a cell full of fluid, while a zero value would indicate that the cell contains no fluid. Cells with F values between zero and one must then contain a free surface. The time dependence of F is governed by the equation:

$$\frac{\partial F}{\partial t}+u\frac{\partial F}{\partial x}+v\frac{\partial F}{\partial y}=0 \tag{4}$$

Boundary Conditions

The problem of propagating waves governed by Eqs.(1), (2) and (3) is considered as a boundary value problem which has three kinds of boundary conditions, namely, the obstacle boundary conditions (at bottoms, walls), the lateral boundary conditions and the free surface boundary conditions.

Except for the special concentration of boundary layer, the free slip boundary condition is adopted which demands zero value of normal velocities on the obstacle surfaces.

$$\vec{\mathbf{u}}\cdot\mathbf{n}=0 \tag{5}$$

The free surface boundary conditions include the kinematic and dynamic boundary condition. In VOF method, the kinematic boundary condition is automatically satisfied by Eq.(4) which means the F function (fluid interface)

Key words: VOF method, submerged reef, nonlinear waves, wave breaking
Department of Civil Engineering, Saitama University, Shimo-Okubo 255, Urawa, Saitama 338

On the offshore boundary , regular waves are generated by surface elevation and velocity components according to various wave conditions and the corresponding wave theories. In the present study, the Stokes third order wave theory is applied.

On the onshore boundary , the incoming waves should leave the calculation domain freely. In most of numerical models, the Sommerfield radiation condition is applied to define this open boundary problem. But recently further study found that applying the Sommerfield radiation condition alone can not fulfill the above mission and mass will accumulate with time which finally leads to a mean water level raise. Accordingly, in the present study, a damping zone(Arai, 1993) and the Sommerfield radiation condition are combined to keep the reflective wave as small as possible.

Iteration Procedure

The basic procedure for advancing a solution through one increment in time, δt, consists of three steps: (1)Explicit approximations of momentum equations are used to compute the first guess for new time-level velocities using the initial conditions or the previous time-level values for all advective values, pressure, and viscous terms. (2)To satisfy the continuity equation, the pressures are iteratively adjusted in each cell and the velocity changes induced by pressure changes are added to the velocities computed in step(1). (3)Finally, the F function defining fluid regions must be updated to give the new fluid configuration. Repetition of these steps will advance a solution through any desired time interval.

Computational Results

In order to verify the numerical model, comparisons are made between the available experimental data and the calculation results . In the calculation, the incident wave conditions are kept the same as that of experiments which are referred in the figures.

Fig.1 shows the sketch of experimental setup. Figs.2 and 3 show the examples of water surface and the horizontal velocity respectively. The comparisons between the calculated and measured results show that the numerical model can give rather accurate results. The surface elevation is shown in Fig.4. From the figure we can see a breaker is initiated at the edge of the reef which is very similar to that in the experiment.

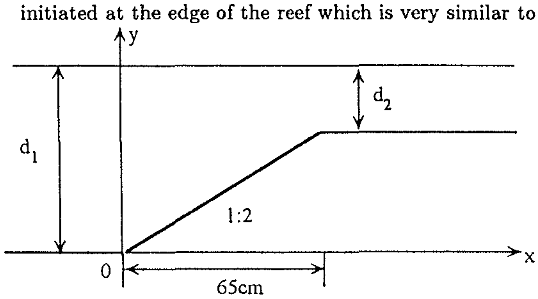

Fig.1 Sketch of experimental set-up

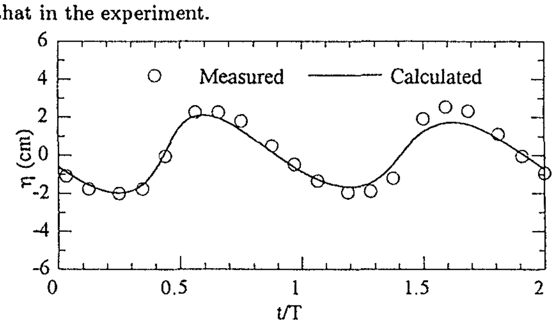

Fig.2 Comparison of water surface at x=85.1cm
(d_1=45.0cm d_2=12.5cm T=1.6s H_{in}=3.3cm)

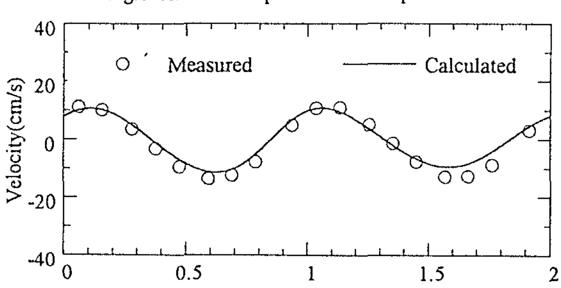

Fig.3 Comparison of horizontal velocity
at x=48.9cm y=33.75cm
(d_1=45.0cm d_2=12.5cm T=1.6s H_{in}=3.3cm)

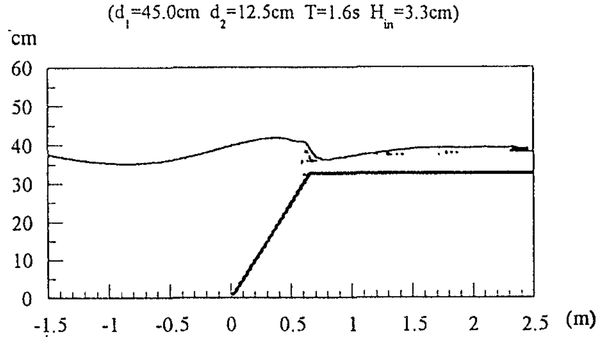

Fig.4 Surface elevation of a braker
(d1=37.5cm d2=5.0cm T=1.6s Hin=5.64cm)

Conclusions

In this study, a numerical model based on the Navier-Stokes equations and the VOF method is applied to simulate wave motion on a submerged reef. The comparisons between the numerical model and the physical experiments show that the numerical model can provide very good accuracy for water surface and particle velocity. The breaker type and breaking position are also right.

References

Hirt,C.W. and Nichlos, B.D.(1981): Volume of fluid method for dynamics of free boundaries. Journal of Computational Physics, Vol.39, pp201-225