THE NUMERICAL SIMULATION OF THE GREEN WATER EFFECT

D.C. Wan[+] and G.X. Wu[++]

[+](Shanghai Institute of Applied Mathematics and Mechanics, Shanghai University, Shanghai 200072, China)

[++](Department of Mechanical Engineering, University College London, London WC1E 7JE, UK)

1 INTRODUCTION

When a ship travels in severe sea conditions or a turret-moored system operates in rough weather, its bow may become immersed in the water. As a result a solid and compact mass of water, or the so called "green water", will wash along and across the deck. During the process, large impact pressures and forces can occur when this block of water which may travel at high speed strikes the obstructions in its way. It has been noted that this impact is one of the main causes of damage to ship superstructures, deck plating, hatches, cargo and equipment.

Although green water loading is becoming an important design issue and has received some attention in the hydrodynamic community in the recent years, little is understood about this complex process. The number of publications for the prediction of green water is relatively limited. Most research has been based on model tests(see references [1] and [2]). There are some works published based on the framework of the potential flow theory to analyse the effect of green water. For example, a fully non-linear panel method based on Laplace's equation has been used to simulate the flow of green water(see reference [3]). In the study, the problem was approximated by a two-dimensional model and some simple ship-bow configurations comprising a vertical wall and a horizontal deck structure (with or without a smooth deck edge) were considered. The results provided include the wave profile. However, the details of the velocity field and the impact pressure were not presented. Although these works have provided some useful results, there are several steps ahead before some better understanding of the green water effect can be achieved. Indeed, in his report to the Health and Safety Executive, UK, Standing (in the reference [4]) has concluded that "there is no reliable theoretical method for predicting the height of water on deck and water particle velocities across the deck."

The main objective of the present study is to make some useful contribution to the numerical solution of this difficult problem. We will base our analysis on the two-dimensional flow and consider a longitudinal section of a typical ship with a tower positioned on the deck. The flow will be governed by the Navier-Stokes equations. The fully nonlinear boundary condition on the free surface will be satisfied at each time step and the evolution of the free surface is achieved by using the Volume-of-Fluid (VOF) method. The analysis allows the wave to overturn and break. It can also capture the interaction of viscous flow with the free surface if the mesh used is of sufficient resolution.

2 FORMULATION AND NUMERICAL METHOD

A coordinate system where Ox coincides with the horizontal bottom and Oy is positive upward is chosen. The horizontal and vertical components of the velocity are given by u and v, while t, ρ, ν, p and g denote time, density, kinematic viscosity of water, pressure and gravitational acceleration, respectively. Surface tension is neglected. The governing equations for the fluid flow are the continuity equation and the momentum equations. For an incompressible fluid they can be written as follows:

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

$$\frac{\partial u}{\partial t}+u\frac{\partial u}{\partial x}+v\frac{\partial u}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\nu\left[\frac{\partial^2 u}{\partial x^2}+\frac{\partial^2 u}{\partial y^2}\right] \tag{2}$$

$$\frac{\partial v}{\partial t}+u\frac{\partial v}{\partial x}+v\frac{\partial v}{\partial y}=-\frac{1}{\rho}\frac{\partial p}{\partial y}-g+\nu\left[\frac{\partial^2 v}{\partial x^2}+\frac{\partial^2 v}{\partial y^2}\right] \tag{3}$$

These equations are solved using a finite difference method. To overcome the difficulty in solving the pressure in incompressible flows, it is desirable to introduce some artificial compressibility effects. Equation (1) is therefore replaced by the following equation:

$$\frac{1}{\rho c^2}\frac{\partial p}{\partial t}+\frac{\partial u}{\partial x}+\frac{\partial v}{\partial y}=0 \tag{4}$$

where c is the artificial speed of sound in the fluid.

The free surface is dealt with by the VOF method [5]. In this method, a function R is defined to have a value of unity at any point occupied by the fluid and a value of zero otherwise. When it is averaged over a cell of the computational mesh, the resulting value F represents the fractional volume of the cell occupied by the fluid. In particular, a unit value of F corresponds to a cell full of fluid, whereas a zero value indicates that the cell contains no fluid. Cells with F values between zero and one contain a free surface. Therefore F is a step function and can be used to define the free surface. Furthermore, the variation of F from cell to cell can be used to define the slope and direction of the interface line in the cell. The time dependence of F is governed by

$$\frac{\partial F}{\partial t}+u\frac{\partial F}{\partial x}+v\frac{\partial F}{\partial y}=0 \tag{5}$$

For the purpose of calculations, the free surface is approximated by a straight line through the cell. The location and slope of the line are determined by the average value of F in the cell and by the gradient of F. We can summarise the calculation procedure at each time step as follows: (a) equations (2) and (3) are solved to obtain the velocities; (b) the pressure in cells within the fluid and on the free surface is adjusted in order that the continuity equation (4) and the free surface dynamic condition can be satisfied; (c) the Donor-Acceptor flux approximation [5]is used to update the value of F; (d) the procedure is repeated, using the updated information from (a)-(c), until convergence has been achieved.

3. COMPUTATIONAL RESULTS

The results of a number of calculations will now be presented to show how a solitary wave evolves into a green water flow on a ship deck. All the calculations were done on a uniform grid with 720 equally-spaced cells($\Delta x=$ 1.0cm) in the x-direction and 60 equally-spaced cells($\Delta y=$ 1.0cm) in the y-direction. The initial shape of the solitary wave and its velocity distribution are given as follows:

$$\eta = H\,\mathrm{sech}^2\left[\frac{K}{D}(x-x_0)\right] \quad K=\left[\frac{3H}{4D}\left(1+\frac{H}{D}\right)\right]^{1/2} \quad U=\frac{S\eta}{D+\eta} \quad S=\left[gD\left(1+\frac{H}{D}\right)\right]$$

where η is the wave elevation and U is the velocity. The other parameters adopted are defined as follows: x_0 is the initial location of the wave crest, H is amplitude of the incoming solitary wave, D is the undisturbed water depth; L and Hs are the total length and height of the ship respectively; Dw is the draft depth; Dd is the distance between the deck and the undisturbed free surface; and h and l

are the height and width of the tower on the deck. For the three test cases considered, H has values 8.0cm, 7.0cm, 6.0cm, respectively; the other parameter values are fixed and are given as follows: x_0 = 140cm , D = 32.0cm, L = 80.0cm, Hs = 15.0 cm , Dw = 2.0cm, Dd = 8.0cm, h = 5.0cm and l = 15.0cm. The gravitational acceleration g is taken as $9.8\,m/s^2$ and the kinematic viscosity has the value $\nu = 1.0 \times 10^{-6}\,m^2/s$. Non-slip body boundary conditions are imposed on the body surface and on the sea bottom. On the left and right boundaries of the computational domain, a first-order open-boundary condition of zero normal gradient is adopted. The Courant-Friedrichs-Lewy criterion, which describes the numerical stability, is used to adjust the maximum time step.

Fig.1 shows the snapshots of the flow fields and wave profiles of the calculated results at several time intervals. In these pictures, the ship is fixed, while the incoming solitary wave moves from left to right. The visualization of the results of this simulation shows water run-up along the front side wall of the ship, passing the deck edge, washing onto the deck, impacting and reflecting on the front wall of the tower, before forming a water jet which flows back into the sea.

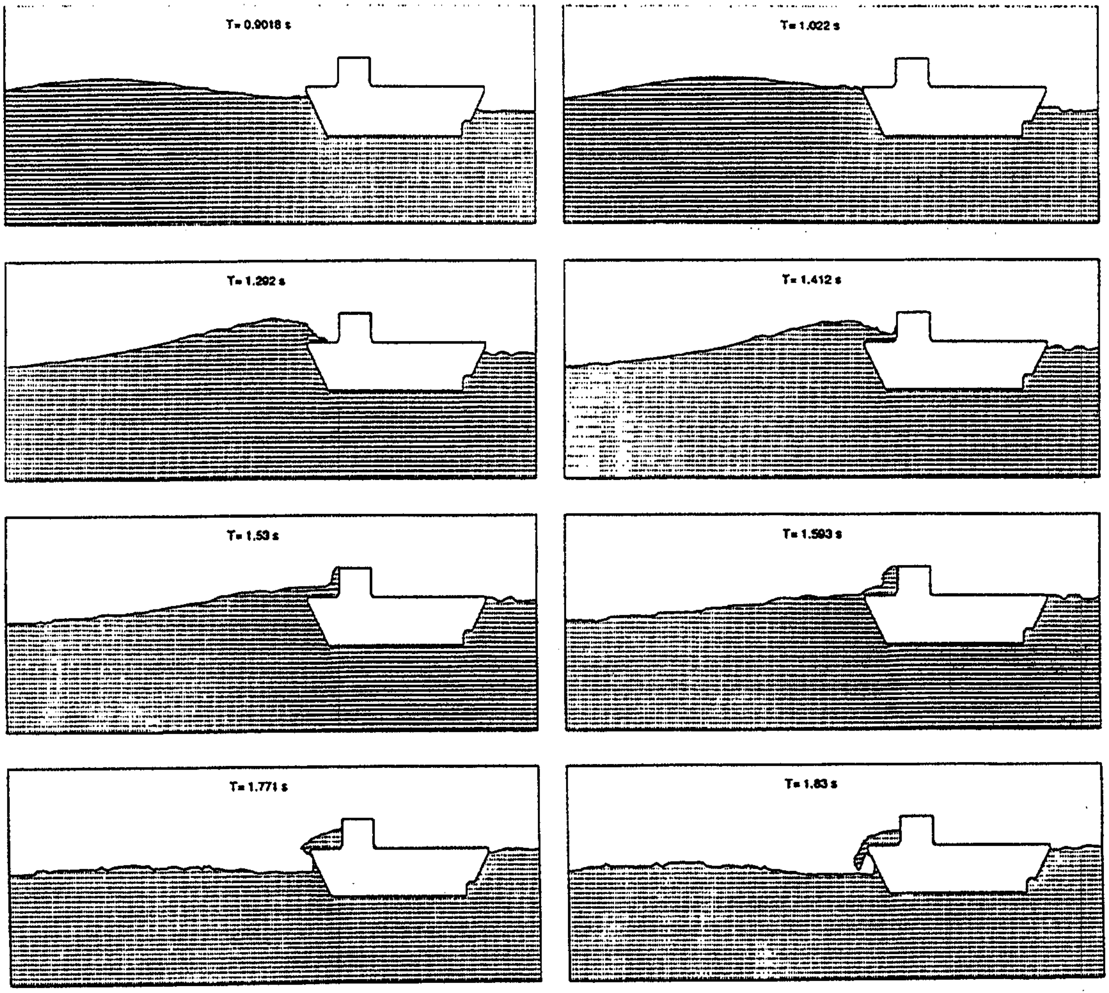


Fig.1 Velocity field and wave profile

Fig.2 shows the velocity vector field and wave profile in a localised area at the front of the ship at two typical moments in time. It can be seen that as soon as the water is on the deck, the water has large velocity in the longitudinal direction and near zero velocity in the vertical direction at the most forward part of the bow, ie, forming water washing along the deck. When hitting the tower, the water is reflected and flow reversal is apparent.

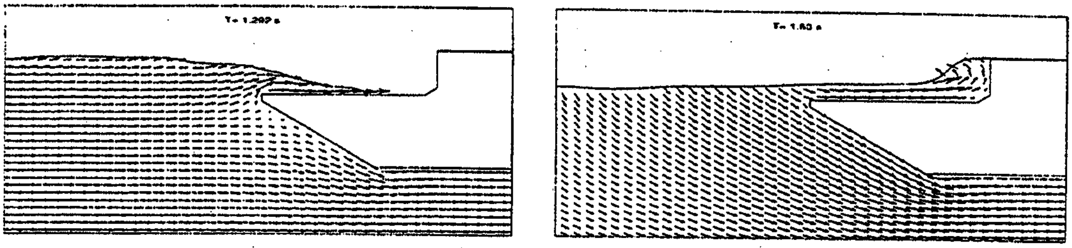
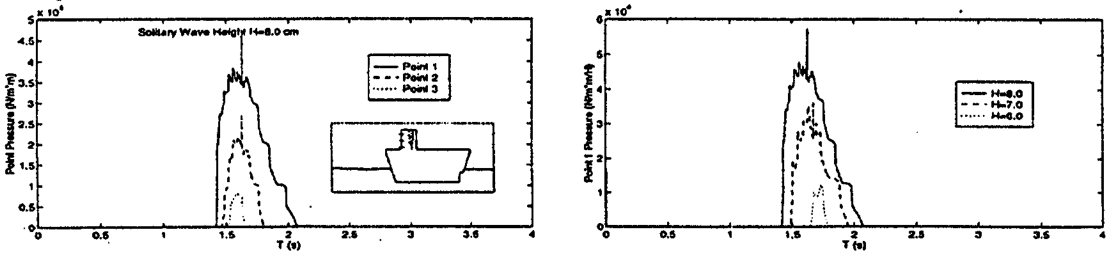


Fig.2 Local velocity vector and wave profile

Fig.3 shows the pressure history on the front wall of the tower. In Fig.3(a), the pressure history comparison between three different points(with vertical distance to deck as 1.0, 2.5, 4.0cm, respectively) with the same incoming solitary wave height $H = 8.0$ is given. It can be seen that the front wall of the tower endures a large impact pressure when the water strikes it, and we find that the largest impact pressure occurs on the corner point, ie, point 1, while the top position of the tower (point 3) has the smallest impact pressure. In Fig.3(b), the point 1 pressure history comparison between three cases with different incoming solitary wave heights $H = 8.0, 7.0, 6.0$cm is given. In the figure, the pressure value has been divided by the value of its incoming solitary wave height. We can see that the higher the incoming solitary wave height, the larger the impact pressure on the front wall of the tower will be. From these figures, it can also be observed that the pressure shows some spikes. Whether this result is physical needs further investigation. From numerical point of view, there are two main issues: firstly, the pressure changes very rapidly during impact so that the time should have been sufficiently small in order to capture the time variation accurately (we did try various different time steps with the smallest $\Delta t = 10^{-5} s$ and found the magnitude and location of the spike vary); secondly, the first-order pressure differential scheme may not be accurate enough, leading to some numerical error for this type of problem of considerable degree of complexity. The behaviour of the impact pressure solution therefore requires further improvement, in particular during water hits the vertical wall of the tower.



(a) (b)

Fig.3 Pressure history: (a) Comparison between different points with the same value of H (equal to 8.0 cm); (b) Comparison between three cases(H = 8.0, 7.0, 6.0 cm) at the same point(point 1).

ACKNOWLEDGEMENT

DCW is supported by the Royal Society through the K.C. Wang Fellowship scheme and GXW is supported by the Royal Society through the Industry Fellow Scheme.

REFERENCES

[1] Buchner B., The impact of green water on FPSO design, Offshore Technology Conf., paper No. OCT 7698, Houston, pp. 45-57, 1995.

[2] Buchner B., The influence of the bow shape of FPSOs on drift forces and green water, Offshore Technology Conf., paper No. OTC 8073, Houston, pp.389-400, 1996.

[3] Buchner, B. and Cozijn, J.L., An investigation into the numerical simulation of green water, Proceedings of the 8th Int. Conference on the Behaviour of Offshore Structures, BOSS'97, Vol. 2, pp.113-125, 1997.

[4] Standing, R.G., Green Seas Damage on FPSOs and FSUs, Health and Safety Executive - Offshore Technology Report, HSE Books, 1997.

[5] Hirt, C.W. and Nichols, B.D., Volume of Fluid(VOF) Method for the Dynamics of Free Boundaries. J Comput Phys, 39, pp.201-225, 1981.