SIMULATION OF GREEN WATER LOADING BY A THREE-DIMENSIONAL TWO-PHASE NUMERICAL MODEL

R. Wemmenhove, G.E. Loots, R. Luppes and A.E.P. Veldman
University of Groningen, Department of Mathematics,
P.O. Box 800, 9700 AV Groningen, The Netherlands
E-mail: r.wemmenhove@math.rug.nl

SUMMARY

This paper describes a study of simulation of green water loading on the deck of a ship. The numerical model has been developed as a one-phase model initially, but has been extended to take two-phase flow effects into account. The method used for the simulations is based on the Navier-Stokes equations and the model is discretised using a finite volume method. Pressure, density and velocity are calculated in the entire flow domain. As a first step, the two-phase flow model considers air as an incompressible second phase. The incompressible two-phase flow model has been validated on a dambreak problem.

1 Introduction

Green water loading on offshore structures is related to violent weather conditions, inducing high and steep waves. Since a large amount of water is flowing on the deck during green water conditions, often causing damage to deck houses or other equipment, there is a great need of simulation tools that can predict the impact loads of green water and give more insight in the local impact phenomena [3]. Green water loading is associated with complex mixtures of air and water, appearing as air pockets, spray and bubble clouds. The physics of the liquid phase are generally considered most important for offshore hydrodynamic problems. Therefore, many mathematical and numerical models only take one phase into account. This approach satisfies until flow effects like wave breaking are getting significant. Considering the second phase (air or another fluid) is useful as soon as the water can no longer be considered as one homogeneous medium. For those conditions, the second phase may affect the water height and pressure level significantly.

2 Physical-mathematical model

In an offshore environment, a one-phase model only considers the flow variables like pressure and velocity in the water phase. The pressure and velocity boundary conditions at the free surface are based on continuity of normal and tangential stresses [3]:

$$-p + 2\mu\frac{\partial u_n}{\partial n} = -p_0 + \sigma\kappa \qquad (1)$$

$$\mu(\frac{\partial u_n}{\partial t} + \frac{\partial u_t}{\partial n}) = 0 \qquad (2)$$

Here, u_n and u_t are the normal and tangential component of the velocity respectively, p is the pressure, p_0 is the atmospheric pressure, σ is the surface tension and κ denotes the total curvature of the free surface.

A two-phase model implies a registration of pressure, velocity and density as well in the second (air) phase, see figure 1. Since these variables are calculated in both phases, the boundary conditions at the free surface ((1) and (2)) are no longer needed in a two-phase model.

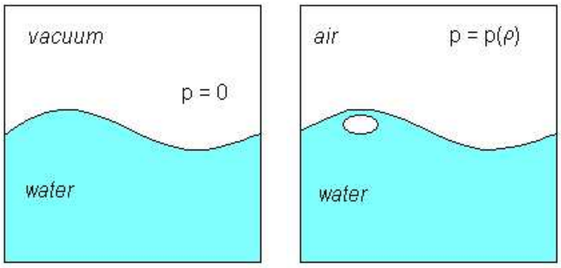

Figure 1: *Calculation domain for one-phase (left) and two-phase (right) model*

The continuity and momentum equation for the two-phase model are

$$\nabla \cdot u = 0 \qquad (3)$$

$$\rho\frac{\partial(u)}{\partial t} + \rho\nabla\cdot(uu) + \nabla p = \nabla\cdot(\mu\nabla u) + \rho F \qquad (4)$$

with density ρ, time t, velocity u, dynamic viscosity μ and F an external body force (such as gravity). For incompressible flow, the density ρ has a constant value in each phase. At locations with a mix of both phases the density is calculated by weighted averaging, while the viscosity is then calculated by harmonic averaging. For some applications it is necessary to consider the compressibility of at least one of the phases. For compressible flow, the pressure p depends on the local density distribution inside both phases. The continuity and momentum equation are now given by

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho u) = 0 \qquad (5)$$

$$\frac{\partial(\rho u)}{\partial t} + \nabla \cdot (\rho uu) + \nabla p = \nabla \cdot (\mu \nabla u) + \rho F \quad (6)$$

The pressure-density relation in the compressible model can be derived by subtracting the time-differentiated continuity equation (5) from the divergence of the momentum equation (6). Neglecting the right-hand side of the momentum equation (6), this gives

$$\nabla^2 p + \nabla^2(\rho uu) - \frac{\partial^2 \rho}{\partial t^2} = 0 \quad (7)$$

Using the speed of sound a, given by $a^2 = dp/d\rho$, the third term in equation (7) can be rewritten as

$$\frac{\partial^2 \rho}{\partial t^2} = \frac{\partial}{\partial t}(\frac{\partial \rho}{\partial t}) = \frac{\partial}{\partial t}(\frac{\partial \rho}{\partial p}\frac{\partial p}{\partial t}) = \frac{\partial}{\partial t}(\frac{1}{a^2}\frac{\partial p}{\partial t}) \quad (8)$$

For a mixture of air and water, the speed of sound is high in pure salt water ($1510\ ms^{-1}$) or pure air ($340\ ms^{-1}$), but much lower (down to $25\ ms^{-1}$) for a mixture of water and air.
Assuming incompressible conditions is related to the assumption of an infinite speed of sound a. Incompressibility of both phases in the numerical model will be assumed for the validation in Section 4.

3 Numerical model

The Volume Of Fluid (VOF) algorithm as developed by Hirt and Nichols [2] is used as a basis for the fluid advection. Compared to level-set methods, a major advantage of VOF methods is their conservation of mass [4]. For the two-phase model, the VOF function determines the filling ratio of the first fluid in each computational cell.
In a one-phase approach, the method solves the incompressible Navier-Stokes equations with a free-surface condition on the free boundary, see eq. (1) and (2). This is in contrast with the two-phase approach, when the VOF function is used to determine the aggregated density value inside a grid cell. The air-water interface is still reconstructed using the VOF function.
The current numerical model, ComFLOW, is an improved 3D Volume Of Fluid (iVOF) Navier-Stokes solver. Initially developed to study the sloshing of liquid propellant in satellites [1], the program is currently able to simulate green water loading, sloshing in LNG and anti-roll tanks, water entry and wave impact loads on fixed structures.

The discretisation of the incompressible Navier-Stokes equations is done on a staggered grid, using an explicit first order Forward Euler Method:

$$\nabla \cdot u^{n+1} = 0 \quad (9)$$

$$\rho^n \frac{u^{n+1} - u^n}{dt} + \rho^n \nabla \cdot (uu)^n + \nabla p^{n+1} = \nabla \cdot (\mu^n \nabla u^n) + \rho^n F^n \quad (10)$$

In these equations, dt is the timestep and $n+1$ and n denote the new and old time level, respectively.
To describe the numerical implications of taking a second phase into account, it is useful to describe the cell labeling briefly. Figure 2 shows the different classes of cells in the two-phase flow model. Geometry

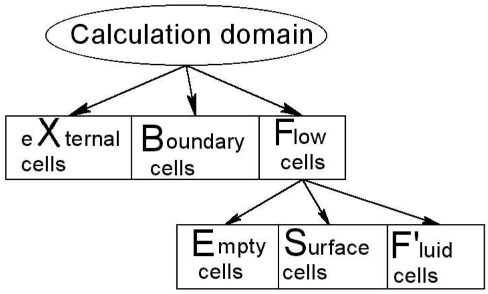

Figure 2: *Schematic overview of cell labeling*

apertures have been introduced to distinguish between solid cells ($F^b = 0$) and cells completely open for flow ($F^b = 1$). Solid cells are classified as eXternal or Boundary cells, while open cells are classified as Flow cells, see figure 2. All cells next to eXternal cells are labeled as Boundary cells, while the remaining cells are Flow cells. For the construction of the interface between the two phases, a VOF function is used which has values $0 \leq F^s \leq F^b$, where F^s is volumetric the part of the cell filled with the first phase. In figure 2, for Empty cells $F^s = 0$. Cells neighbouring the Empty cells are labeled as Surface cells, while the remaining cells are Fluid cells. Due to this labeling method, the Fluid cells are partially or completely filled with the first (liquid) phase.
The labeling in figure 2 is equal to the labeling in the one-phase model. However, the velocity and pressure are now calculated as well in the Empty cells, which are completely filled with air. Furthermore, the viscosity and density are now variable in space, so they are registrated in all Flow cells.

4 Model results

The two-phase numerical model has been tested on several offshore problems, just as the one-phase model. One of the test cases is the dambreak simulation, which can be regarded as a simple model of green water flow on the deck of a ship. The numerical simulation is compared with model experiments performed at Maritime Research Institute Netherlands (MARIN). During the experiment measurements have been performed of water heights, pressures and forces on different locations. The small box in figure 3, which represents a container, is covered by eight pressure sensors, while the water height is measured at several locations behind and in front of the small box. Figure 3 shows snapshots of the model experiment and the numerical simulation in an early stage of the experiment, just before the water front reaches the left wall.

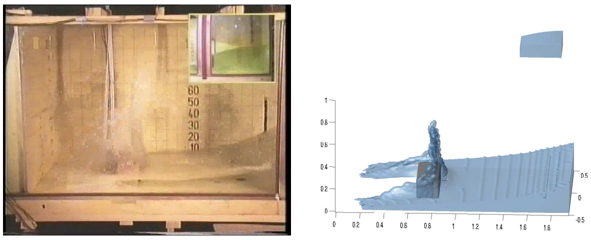

Figure 3: *Snapshots of dambreak experiment and simulation with a box in the flow, t = 0.56s*

Regarding figure 3, there is a visual agreement between the snapshots of simulation and experiment.

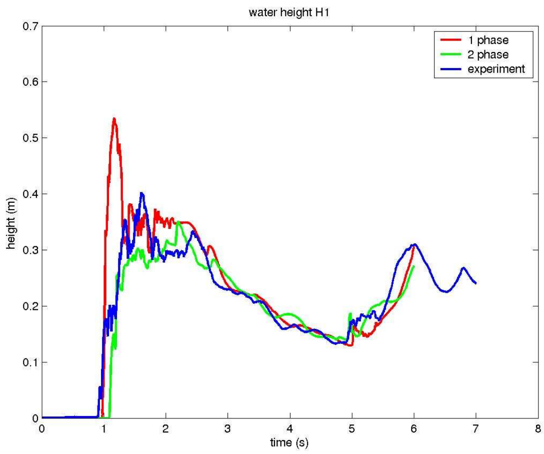

Figure 4: *Water height development just behind the small box*

Figure 4 shows the water height development just behind the small box. The one-phase model predicts a water level at $t = 1s$ which is clearly too high, while the rise time is too short. The results of the experiment and the two-phase model show a large resemblance, although the water in the two-phase simulation reaches the measurement location slightly later. This may be caused by the treatment of the air viscosity in the model. Due to the upwind discretisation, artificial diffusion in the model changes the viscosity of the air to a value larger than the physical one. Therefore, the water needs a larger effort to push the air away, resulting in a slower advancement of the water front in the numerical model.

For the pressure level development (figure 5), the overall trend of measurement and numerical simulations is quite similiar. The initial pressure peak

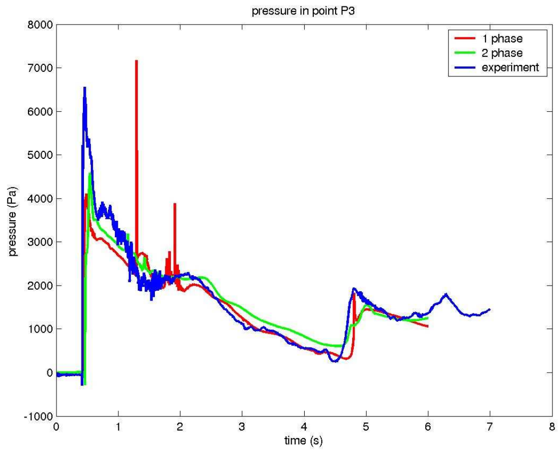

Figure 5: *Pressure development at the front of the small box*

of the experiment is not correctly predicted by the one-phase simulation. The correlation between the two-phase simulation and the experimental results is better. Furthermore, the one-phase simulation shows some artificial pressure spikes that are not present in the two-phase simulation. The largest of these artificial pressure spikes, at $t = 1.32s$ (see figure 5), will be considered in more detail.
Figure 6 shows the pressure values around the small box (see figure 3), one timestep before and at the moment of the pressure spike.

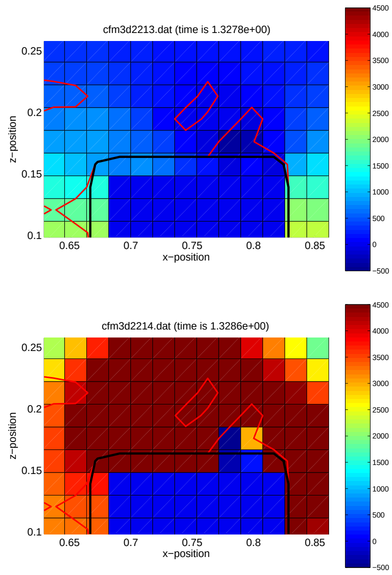

Figure 6: *One-phase model - pressure field around the small box in figure 3. The peak pressure values are generated one timestep after the changes in cell labeling shown in figure 7.*

The pressure spikes in the one-phase model originate from the cell labeling method as described in figure 2. Due to the violent fluid motion, many topological changes may occur simultaneously within a single timestep in the numerical simulation. Figure 7 shows

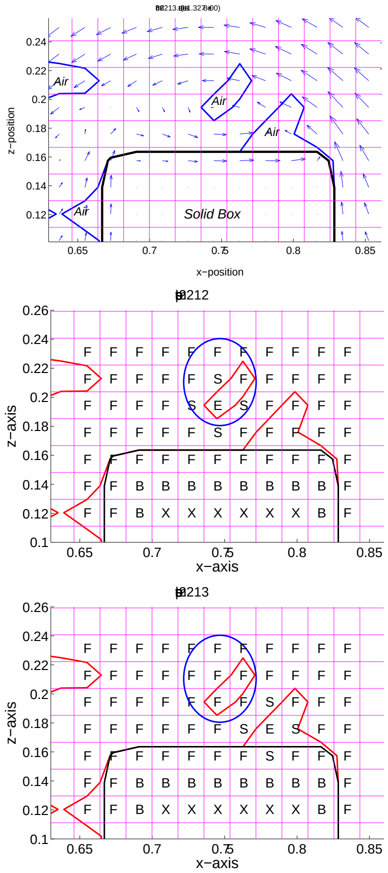

Figure 7: *One-phase model - a) velocity field and b,c) cell labeling around the small box in figure 3. The pressure development shown in figure 6 is induced by rapid changes in cell labeling. Between timesteps 2212 and 2213, the Empty Cell in the air pocket* $(x = 0.75, y = 0.2)$ *changes to a Surface cell, causing an artificial pressure spike around this air pocket.*

the velocity field and cell labeling just before the highest pressure spike in the one-phase simulation (at $t = 1.32s$). For the Empty and Surface cells in the one-phase model, mass conservation is not required by the numerical algorithm. Due to the rapid transition from Empty cells and Surface cells to Fluid cells, the pressure has to 'work' to achieve mass conservation in the newly created Fluid cells. This 'work' will manifest itself in a spike in the pressure signal of the one-phase model.

For the two-phase model, mass conservation is also applied to the Empty and Surface Cells. Mass conservation in all Flow cells (see figure 2) prevents the model from giving pressure spikes as a result of changing cell labels. The result is a smoother pressure signal of the two-phase model, as shown in figure 5.

Compared with other test cases in offshore environments, large air pockets, for example occuring during wave slamming, are quite insignificant for the dambreak problem. As soon as air pockets are present more frequently, including compressibility of the air phase may be valuable.

ACKNOWLEDGEMENTS

This research is supported by the Dutch Technology Foundation STW, applied science division of NWO and the technology programme of the Ministry of Economic Affairs. MARIN is greatly acknowledged for providing the measurement data of the dambreak problem.

References

1. Gerrits, J., *Dynamics of Liquid-Filled Spacecraft - Numerical Simulation of Coupled Solid-Liquid Dynamics*, PhD thesis, University of Groningen, 2001.
2. Hirt, C.R., Nichols, B.D., *Volume of Fluid (VOF) method for the dynamics of free boundaries*, Journal of Computational Physics, **39**, 201-225, 1981.
3. Kleefsman, K.M.T., Fekken, G., Veldman, A.E.P., Iwanowski, B., Buchner, B., *A Volume-Of-Fluid Based Simulation Method For Wave Impact Problems*, accepted for publication by Journal of Computational Physics, 2005.
4. Sethian, J.A., Smereka, P., *Level Set Methods for Fluid Interfaces*, Annual Review of Fluid Mechanics, **35**, 341-372, 2003.