

**PREDICTION OF GREEN WATER AND WAVE LOADING USING A NAVIER-STOKES
BASED SIMULATION TOOL**

**K.M.Theresa Kleefsman
Geert Fekken
Arthur E.P. Veldman**

Institute of Mathematics and Computing Science
University of Groningen
PO Box 800
9700 AV Groningen
The Netherlands
Email: theresa@math.rug.nl

**Tim H.J. Bunnik
Bas Buchner**

MARIN
PO Box 28
6700 AA Wageningen
The Netherlands

Bogdan Iwanowski
CorrOcean ASA
Claude Monets alle 5
1338 Sandvika
Norway

ABSTRACT

Results of computer simulation of wave and green water loading on floating offshore structures are presented. The simulation program used is a CFD code which solves the Navier-Stokes equations that describe flow of incompressible viscous fluids. The Navier-Stokes equations are discretised using a Finite Volume method on a Cartesian grid with staggered variables. The free surface is displaced using a Volume Of Fluid based algorithm combined with a local height function. In this paper results of validation and sensitivity tests of simulation of green water on the foredeck of an FPSO are presented. Here, the waves are modeled as a dam of water around the deck which is suddenly released. Furthermore, wave loading from impact of regular waves on a SPAR platform is computed and compared with experimental results. The program is found to be robust and the computational results show good agreement with the experiments.

INTRODUCTION

Ever since the beginning of the oil industry at sea, the offshore industry was challenged to explore and extract oil in water of increasing depth. Therefore, an increasing use is made of floating systems like Floating Production Storage and Offloading (FPSO) systems. After such a floating system has come into use,

it usually stays at the same location for 20 years or more. Special design requirements are needed for such a system. Over the last few years, several incidents have shown that these floating systems suffer from green water on the deck and impact from waves at the bow, like slap or slam events. An example of a damage incident is the wave impact by a relatively small but steep wave at the Schiehallion FPSO which is located in 395 m deep water to the West of the Shetland Isles (Gorf, 2000).

In January 2001 a project sponsored by the offshore industry and the European Union has been started to develop risk assessment procedures and guidance tools to improve safety, reliability and availability of floating offshore structures. In this project, the focus is on extreme events with loads that should be included in the design of FPSO systems. As part of the project a calculation method will be developed to predict green water and wave impact loading. This method is incorporated in the CFD tool ComFLOW which is developed at the University of Groningen. The ComFLOW program is based on the incompressible Navier-Stokes equations that describe the motion of a viscous fluid. Originally, ComFLOW has been developed to simulate the coupled liquid-solid dynamics of spacecraft in a micro gravity environment, see (Gerrits, 2001). Besides that, ComFLOW has already been adopted in several areas, like the medical research by simulating blood flow through vanes and in the maritime industry where water flow in stabilising anti-roll tanks was

simulated (Van Daalen, 2000). At the moment, the possibilities of ComFLOW are being enlarged by including moving objects. Once this has been incorporated, ComFLOW will be able to simulate for example a moving ship at sea.

In this paper, results on the prediction of green water on the foredeck of an FPSO and wave loading on a SPAR platform are presented. In the green water simulations, a wall of water is placed around a fixed ship which is suddenly released. Then, the water starts to flow onto the deck where it hits a structure on the deck. First, results are compared to an experiment performed at MARIN. Then, robustness of the program is tested by doing a number of sensitivity checks. These sensitivity tests consist of grid refinement and some variations in the size of the structure on the deck. In the second tests, simulations of regular wave loading on a fixed SPAR platform are compared to experiments performed at MARIN. The results of these simulations give a first idea of prediction of loading coming from waves.

MATHEMATICAL MODEL

The motion of fluid flow is described by the Navier-Stokes equations coming from conservation of mass and momentum. Applying these conservation laws to a volume V (partially) filled with incompressible, viscous fluid, gives the following equations:

$$\oint_{\partial V} u \cdot n dS = 0, \quad (1)$$

$$\int_V \frac{\partial u}{\partial t} dV + \oint_{\partial V} uu^T \cdot n dS = -\frac{1}{\rho} \oint_{\partial V} (p - \mu \nabla u) \cdot n dS + \int_V F dV. \quad (2)$$

Here, ∂V is the boundary of volume V , $u = (u, v, w)$ is the velocity in the three coordinate directions, n is the normal of volume V , ρ denotes density, p is the pressure, ∇ is the gradient operator. Further μ denotes dynamic viscosity and $F = (F_x, F_y, F_z)$ is an external body force, for example gravity.

Solid wall

Boundary conditions are required for the solid walls, the free surface and the in- and outflow boundaries. On the solid walls, either a no-slip condition $u = 0$ or a free-slip condition $u \cdot n = 0$ and $\frac{\partial u_t}{\partial n} = 0$ is prescribed. Here, n is the normal on the wall and u_t is the component of the velocity tangential to the wall.

Free surface

At the free surface, continuity of normal and tangential stresses is demanded, which can be described by the equations

$$-p + 2\mu \frac{\partial u_n}{\partial n} = -p_0 + 2\gamma H \quad (3)$$

$$\mu \left(\frac{\partial u_n}{\partial t} + \frac{\partial u_t}{\partial n} \right) = 0. \quad (4)$$

Here, u_n is the normal component of the velocity, p_0 is the atmospheric pressure, γ is the surface tension and $2H$ denotes the total curvature. Furthermore, an equation is required for the displacement of the free surface. Suppose the position of the free surface is given by $s(x, t) = 0$, then the motion of the free surface is given by

$$\frac{Ds}{Dt} = \frac{\partial s}{\partial t} + (u \cdot \nabla)s = 0. \quad (5)$$

In- and outflow boundaries

At the inflow boundary, velocities are prescribed from a given velocity profile. To generate a regular wave pattern, Airy wave theory is used to determine velocities and wave height on the inflow boundary.

At the outflow boundary, two different conditions are used. The first one reads $\frac{\partial u}{\partial n} = 0$ and works fine when there is a uniform flow. When the flow has a wave-like pattern, a Sommerfeld based boundary condition is applied which is explained in the next section.

NUMERICAL MODEL

To perform numerical simulations, the flow domain is covered with a Cartesian grid. The exact flow domain is determined by introducing a volume aperture F^b and edge apertures, A^x , A^y and A^z , that indicate which part of the cell volume and cell faces respectively is open to flow. In every cell of the grid that contains fluid, the discretised Navier-Stokes equations will be solved. Therefore, information is required about the nature of the cell: is it a fluid cell, a cell at the free surface, an empty cell or a boundary cell. To indicate this the cells are labeled using a color function F^s which is called the Volume Of Fluid (VOF) function. The volume apertures for the geometry and the VOF function are related as $0 \leq F^s \leq F^b \leq 1$. Cells with $F^b = 0$ are labeled as boundary (B) cells. Interior cells containing no fluid, i.e. $F^b > 0$ and $F^s = 0$ are called empty (E) cells. Cells that contain fluid and are adjacent to empty cells are labeled as surface (S) cells. The other cells are labeled as fluid (F) cells. In Figure 1 an example of the labeling is given.

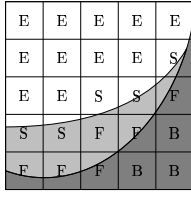


Figure 1. Labeling: dark grey denotes solid body, light grey is liquid

Discretisation of the Navier-Stokes equations

The Navier-Stokes equations are discretised using a finite volume method which starts from Equations 1 and 2. The discretisation is performed on a staggered grid, i.e. the pressure is defined in cell centers whereas the velocities are defined at cell faces.

As an example of discretisation in cut cells, the discretisation of the continuity equation will be explained in detail. Consider a computational cell, as shown in Figure 2, with part of it solid body and the other part liquid. Discrete conservation of

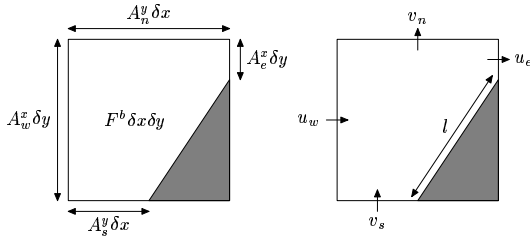


Figure 2. Conservation cell for discretisation of continuity equation

mass implies that the sum of the mass fluxes through the boundary of the conservation cell V equals zero. This results in

$$u_e A_e^x \delta y + v_n A_n^y \delta x - u_w A_w^x \delta y - v_s A_s^y \delta x + 0 \cdot l = 0 \quad (6)$$

where the notation is explained in Figure 2. Because the mass flux through a solid boundary is always zero, Equation 6 can be applied to all computational cells.

The discretisation of the momentum Equations (2) is performed in the same manner, but it would go too far to explain the details in this paper.

For the temporal discretisation of the Navier-Stokes equations the explicit forward Euler method is exploited. This results in the following equations:

$$M u_h^{n+1} = 0, \quad (7)$$

$$\Omega \frac{u_h^{n+1} - u_h^n}{\delta t} + C(u_h^n) u_h^n = -\frac{1}{\rho} (M^T p_h^{n+1} - \mu D u_h^n) + F_h^n. \quad (8)$$

Here, the time level is denoted by the superscript and M , Ω , C and D are coefficient matrices coming from the spatial discretisation. The continuity equation is discretised on a new time level to ensure that the velocity field is divergence free at that time level.

To determine the pressure and velocities at the free surface, the boundary conditions given by Equations 3 and 4 are discretised. In Equation 3 the term $2\mu \frac{\partial u}{\partial n}$ is neglected because this term is small compared to the other terms in the equation. From this equation, the pressure at the free surface equals $p_{fs} = p_0 - 2\gamma H$. Using this, the pressure in surface cells is calculated as an interpolation of the pressure at the free surface and the pressure in an adjacent fluid cell. A velocity at the interface of an S-cell and an E-cell is calculated by demanding mass conservation in S-cells. Sometimes, velocities at the interface of two E-cells are required because they are used in the discretised momentum equations. These velocities are calculated using a simplified version of Equation 4. For a more detailed overview of the discretisation of the Navier-Stokes equations we would like to refer to (Gerrits, 2001).

Outflow boundary condition

At the open outflow boundaries, either a Neumann or a Sommerfeld based boundary condition are used. In this section, the Sommerfeld condition will be explained. On open boundaries, a condition is needed which gives a continuation of the flow with a minimum of reflections into the domain. These reflections do easily occur when the velocities of the fluid inside the domain do not fit nicely to the velocities obtained at the outflow boundary. A Sommerfeld based condition is used which is a simple mathematical continuation having the form of out-going waves. For an outflow boundary in the positive x -direction, the Sommerfeld condition starts from the wave equation

$$\frac{\partial u}{\partial t} + c \frac{\partial u}{\partial x} = 0 \quad (9)$$

where u can be any quantity. To use this equation as an outflow condition, we must apply it to the flow variables at every location on the outflow boundary. After the idea of Orlandi (Orlandi, 1976) this condition is used twice: first, the phase speed c is determined which is then used to calculate the new boundary values. To make this possible, the assumption is made that the same phase speed applies at the boundary and at one cell away

from the boundary. The wave equation is discretised as follows:

$$\frac{1}{2} \left(\frac{u_i^{n+1} - u_i^n}{\delta t} + \frac{u_{i-1}^{n+1} - u_{i-1}^n}{\delta t} \right) + c \frac{1}{2} \left(\frac{u_i^{n+1} - u_{i-1}^{n+1}}{\delta x} + \frac{u_i^n - u_{i-1}^n}{\delta x} \right) = 0 \quad (10)$$

where i is the boundary grid point, $n + 1$ is the last known time level, δt is the time step and δx is the grid size. The temporal discretisation is taken at place $i - \frac{1}{2}$ and the spatial discretisation at time level $n + \frac{1}{2}$. Rearranging this expression gives

$$u_i^{n+1} = \frac{(u_i^n - u_{i-1}^{n+1} + u_{i-1}^n - c\delta t/\delta x(-u_{i-1}^{n+1} + u_i^n - u_{i-1}^n))}{1 + c\delta t/\delta x} \quad (11)$$

To determine the phase speed c , Equation 10 is adopted one grid node to the left (by replacing i by $i - 1$), where all values are known. After that, the phase speed is used to determine the new boundary value u_i^{n+1} from Equation 11. It is assumed that at the outflow boundary only outgoing waves occur. This restricts the phase speed to $c > 0$. Because of stability requirements of the numerical scheme, the phase speed is also restricted by $c < \frac{\delta x}{\delta t}$.

Solution method

To explain the solution method, the terms in Equation 8 are rearranged to

$$u_h^{n+1} = \tilde{u}_h^n + \delta t \Omega^{-1} \frac{1}{\rho} M^T p_h^{n+1}, \quad (12)$$

where

$$\tilde{u}_h^n = u_h^n - \delta t \Omega^{-1} (C(u_h^n) u_h^n - \frac{\mu}{\rho} Du_h^n - F_h^n) \quad (13)$$

First, a temporal velocity field \tilde{u}_h^n is determined using Equation 13. Next, Equation 12 is substituted in Equation 7 which results in a Poisson equation for the pressure. From this equation the pressure is solved using the SOR (Successive Over Relaxation) method where the optimal relaxation parameter is determined during the iterations (Botta, 1985). Once the pressure field is known, the temporal velocity field \tilde{u}_h^n is adjusted using the pressure gradient to a new velocity field.

Free surface displacement

Once the velocity field is known, the free surface can be displaced using an donor-acceptor method which is based on the Volume of Fluid method (Hirt, 1981). In the adopted method, the free surface is reconstructed using a piecewise constant reconstruction where the fluid surface is aligned with one of the

coordinate axes. The fluxes between cells are computed by velocity times the area of the cell, after which they are used to calculate the new fluid apertures F_s . When this method is adopted without further adjustments, flotsam and jetsam that are small bits of fluid which get separated from the fluid, appear in the neighborhood of the free surface. This must be avoided to have a good working method. Therefore, additional to the original VOF method, in free-surface cells a local height function is adopted which displaces the height of the column or row corresponding with the surface cell (Gerrits, 2001). Once the new fluid configuration is known, the free surface labeling can be adjusted.

The CFL condition

For stability, the time step is adjusted during the computation using the CFL-condition. Therefore, the Courant-Friedrichs-Levy (CFL) number is introduced:

$$CFL = \frac{|u|\delta t}{\delta x} + \frac{|v|\delta t}{\delta y} + \frac{|w|\delta t}{\delta z}. \quad (14)$$

For stability of the Forward Euler discretisation method the CFL-number must satisfy the condition $CFL \leq 1$. During the simulation the time step is automatically adjusted using user defined restrictions on the CFL-number.

GREEN WATER LOADING ON AN FPSO

In this section results of simulations of green water on the foredeck of an FPSO will be presented. Green water on the deck of a ship occurs because of a complex interaction of the incoming waves and the motion of the ship. To get a first idea of the possibilities of the numerical method employed by ComFLOW the problem has been rigorously simplified. Firstly, the ship is fixed. Secondly, the incoming waves are modeled as a wall of water standing around the deck of the vessel. Then the water is released as in a dambreak problem and starts to flow onto the deck. To check the performance of the numerical method in capturing the physics of the problem, some tests have been done. First, the code has been validated using model tests performed at MARIN. Next, the sensitivity of the method to the grid size and to small variations in the geometry has been tested.

Validation against experiments

To be able to compare the impact loading of the green water on a deck structure with model tests performed at MARIN, the initial condition has to be tuned to the conditions in the experiment. This is done by a comparison of the contours of the waterfront in the model tests with the contours in the simulation. The model test which has been used for the validation is a regular wave test on a vessel with a bow flare of 30 degrees and

deck width of 47 m. The waves were characterised by a wave amplitude of 8.65 m and wavelength/ship length = $\frac{\lambda}{L} = 0.75$. On the deck of the vessel, a vertical structure is placed which can be looked at as a deck house. See (Buchner, 1995) for more details on the model tests. Using the dambreak modeling of the waves, a rather good approximation of the conditions in this test appeared to be a vertical wall of water around the deck of the vessel which was 13 meter high at the most forward point of the bow and decreased linearly to 5 meter below the deck level behind this point as shown in Figure 3.

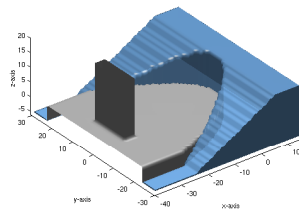


Figure 3. Initial fluid configuration

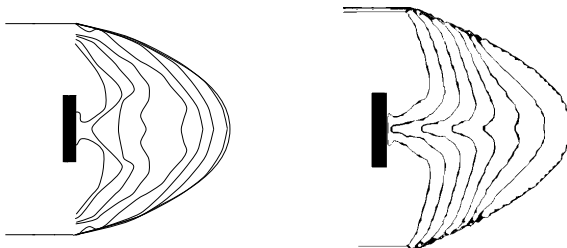


Figure 4. Contours of the water front; left: model test, right: ComFLOW

In Figure 4 the contours of the waterfront at the deck of the simulation and the model test with a time step of 0.31 s are compared. From the contour lines in both the experiment and the simulation, we see that the water on the deck forms a high velocity water tongue which smashes on the rectangular deck structure. In the simulation this water tongue looks a bit sharper. This is only due to a very thin layer of water, so this has not much influence on the impact. The global behavior of the water on the deck is similar in simulation and model test, so we conclude that the dambreak initial condition is a reasonable approximation for the conditions in the test.

During the experiment, some measurements have been performed, which can be used for the validation of ComFLOW. First, at three places on the centerline of the deck (H1, H2 and H3), the water height has been measured and at one place (Pdeck) the pressure has been measured. Furthermore, the wave forces

on the rectangular deck structure have been measured and four pressure panels have been placed on the front of the structure (see Figure 5 for an overview). The measured water height and pressure at the deck has been compared with the water height and pressure calculated by ComFLOW in Figure 6.

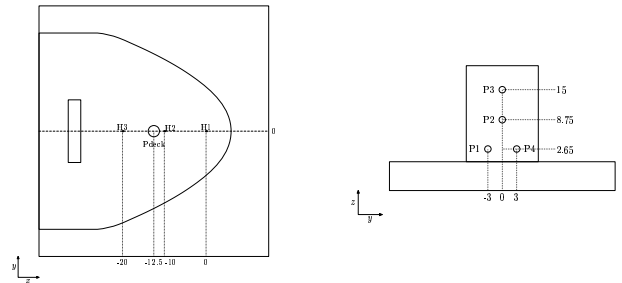


Figure 5. Measurements on the deck and the deck structure

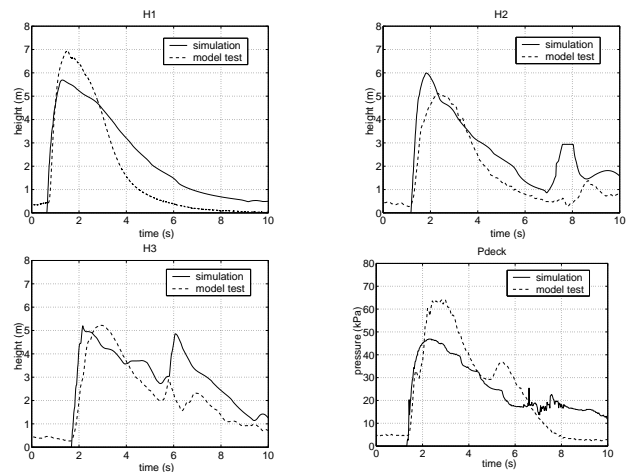


Figure 6. Water height and pressure on the centerline of the deck

From this figure we see that the water at H1, H2 and H3 rises very rapidly, which is well predicted by ComFLOW. The water height at positions H2 and H3 has a second peak which occurs due to the water that has smashed onto the structure and flows back again. These second peaks are also predicted by ComFLOW albeit not at the same time and with the same amplitude. The time trace of the pressure at the deck shows a similar behavior as the water height on the deck. After a short rise time, the pressure slowly decreases. Since the ship is fixed in the simulation, the pressure on the deck calculated by ComFLOW only exists of the hydrostatic pressure. That is why the amplitude of the pressure in the model test and the simulation do not have the same magnitude.

Finally, the wave loads on the deck structure calculated by ComFLOW can be compared with measurements. We will only show the results of the horizontal force and the pressure time series of panel 1 and panel 2 (see Figure 5). The time series of panel 4 are comparable with the series of 1, and the series of panel 3 is not very interesting, because the water does not reach this panel. In Figure 7 the results are presented. As can be seen, almost no water reaches panel 2, both in the model test and in the simulation. Panel 1 is hit by a large amount of water. The resemblance between the model test and the simulation is large. Both reach the maximum pressure at the same time after which a second peak occurs. The same behavior has been found in the total horizontal force.

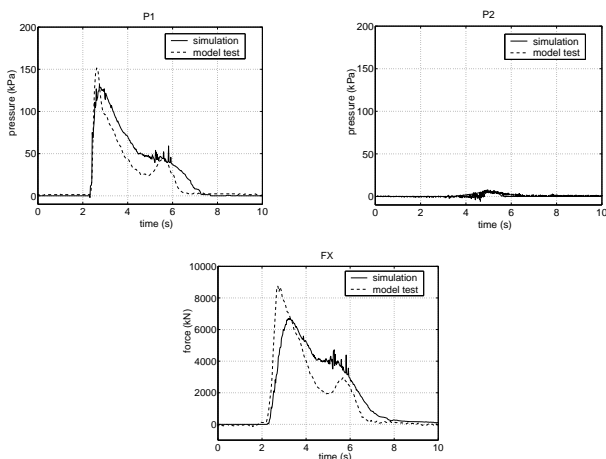


Figure 7. Pressure time series and total horizontal force on the structure

Recapitulating the results of the validation case, we can conclude that the global behavior of the water is well predicted by ComFLOW. Some differences can be seen, especially in the amplitude of the pressure time series and horizontal force. There are several possible explanations for this difference. First, it should be noted that the impact loads are sensitive to variations in the amount of water and velocity of the water. When using waves in stead of a dambreak of a water wall these quantities will have different values. Further, the simulations are simplified by using a fixed ship instead of a moving one, which certainly has had an influence on the resulting loads. Finally, the pressure time series of the simulation are taken in cell-centers of the cells adjacent to the geometry instead of at the geometry. This may give some different values and should probably be improved in the future by extrapolation of the pressure.

Grid refinement study

The accuracy of the simulation results depends on the size of the grid. A finer grid naturally gives more accurate results. To show the dependency of the green water simulation to the size of the grid, a grid refinement study has been performed. For these tests, similar geometry and liquid configuration are used as in the test described above. Calculations have been performed on three different meshes:

Mesh	Cells x	Cells y	Cells z	Total nr. of cells
Coarse	66	72	32	191,520
Medium	100	110	50	640,224
Fine	136	148	68	1,532,160

The results of the total horizontal force on the deck structure are presented in Figure 8. It can be concluded that the results are

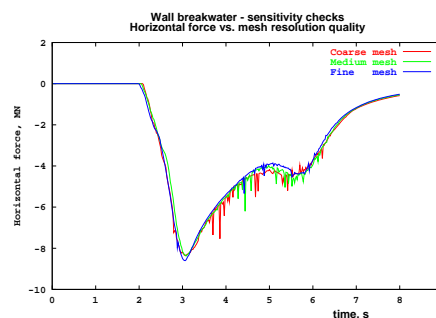


Figure 8. Total horizontal force calculated with three different grids

equivalent for the three different grids. The coarsest grid is fine enough for this kind of simulations. If a fine enough grid is used, the simulation is insensitive to changes in the grid size. In the signal of the force, spikes are present that are due to the handling of surface cells. Most of the causes of spikes are already removed in the newest version of ComFLOW as can be seen in the results presented in the next section.

Sensitivity to geometrical changes

To see how ComFLOW reacts on small changes in the geometry, simulations have been performed of green water on the deck equipped with a cylinder the diameter of which will be altered. A robust computer program should react smoothly on small changes. The diameter of the cylinder on the deck will be set on 6, 12, 18 and 24 meter. In Figure 9 a movie is shown of green water on the deck equipped with a 6 m diameter cylinder. In this movie, the high velocity water tongue is clearly visible. After that, the water impacts the cylinder and starts to flow around it. In Figure 10 the impact loads on the cylinders with the

four different diameters are shown. The results are as expected: the load on the cylinder increases with the diameter of the cylinder. It can be concluded that ComFLOW reacts smoothly on small changes in the geometry.

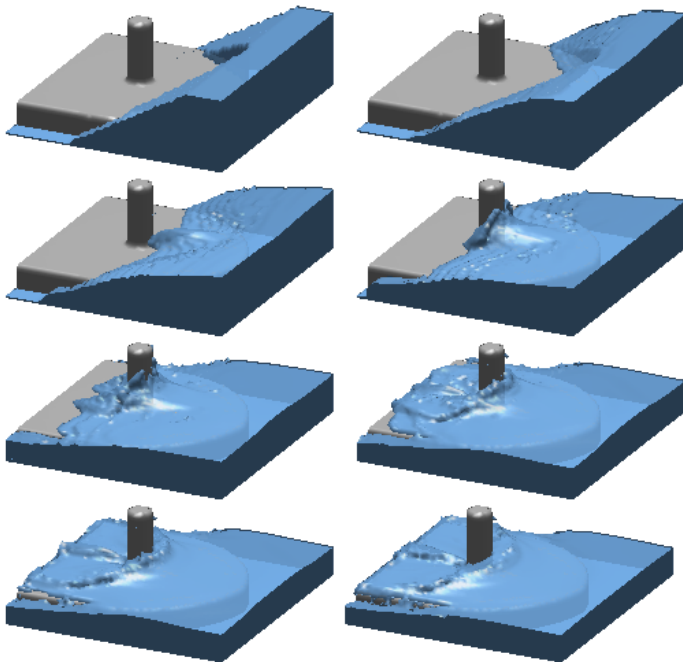


Figure 9. Snapshots of a green water simulation, time step 0.8 s

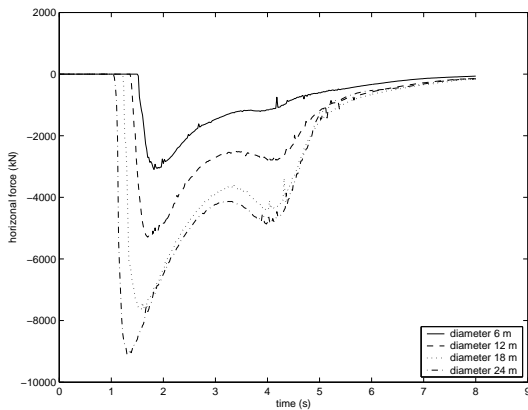


Figure 10. Total horizontal force on cylinders with altered diameter

WAVE LOADING ON A SPAR BUOY

To validate the program on the issue of wave loading, simulations have been performed of regular wave loading on a SPAR

Buoy. At MARIN, experiments with a SPAR have been done where the SPAR was fixed while regular waves hit the structure. In full scale, the SPAR has a total length of 220 m and a diameter of 35 m, the draft is 200 m in a water depth of 290.35 m. The SPAR has been divided into three horizontal segments on which forces have been measured (see Figure 11). The characteristics

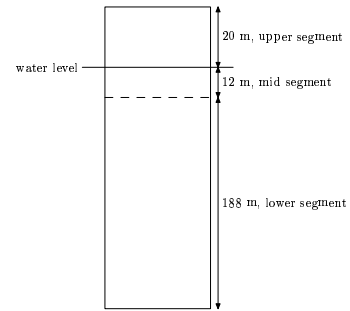


Figure 11. Configuration of the SPAR buoy

of the waves used in the simulation which has been performed are given in the table below. The direction of the waves is 270 degrees.

Test No.	frequency	period	amplitude	wave length
101	0.26 rad/s	24.17 s	5.383 m	866 m

The SPAR buoy has been placed on the distance of one wave length behind the inflow boundary. Behind the cylinder some extra meters are added to the domain before the water flows out through the outflow boundary. On the inflow boundary, the left wall of the domain, the velocity profile and water height have been prescribed using Airy wave theory.

The simulations have been performed on a grid with about 100 cells in the wave length, 20 cells in the transverse direction and 60 cells in the total height of the domain. The grid is stretched in the z-direction towards the free surface. In Figure 12 the calculated water height at 290 meter in front of the buoy has been compared to the measured water height. The water in front of the buoy increases during the simulation, which is also the case with the total amount of water in the numerical domain. This happens because the water does not perfectly flow out of the domain at the outflow boundary on the right. This same observation can also be made when looking at the pressure at the fore side of the buoy 18 meter below the water level as is shown in the right part of Figure 12. The mean level of the pressure increases in time because the water level has increased.

A comparison of the calculated and measured forces in the direction of the waves on the SPAR buoy is presented in Figure 13. The calculated and measured total force on the structure match very well. The predicted maximum of the force on the

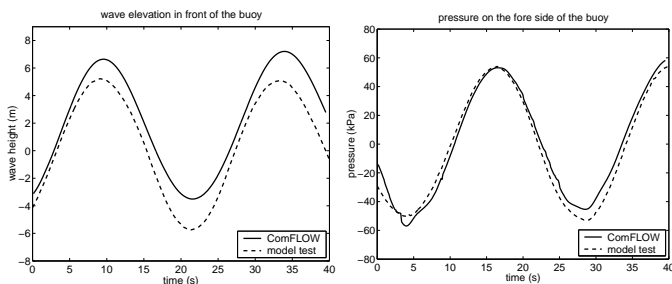


Figure 12. Test 101: wave elevation (left) and pressure on buoy (right)

upper segment is larger than the measured one. More water is flowing around the upper segment because of the increased water level. In the forces some peaks can be observed. Partly, they are due to pressure in empty cells that changed to a fluid cell in one time step (for example the peak at about 24 seconds at the mid segment). At the lower segment some other peaks can be observed (also at time is 24 seconds) that have influence in more than one time step. These peaks are due to the treatment of the outflow boundary. More investigation is required to solve this problem.

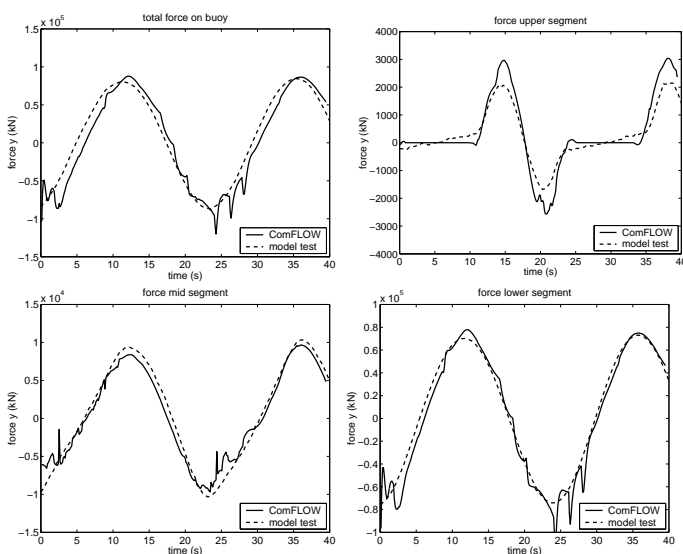


Figure 13. Test 101: forces on the total structure and the segments

CONCLUSIONS

The simulation program ComFLOW which solves the Navier-Stokes equations has been validated using experiments and it has been checked on sensitivity to changes in grid and geometry. Looking at the results of green water flow on the deck of

a vessel, it can be concluded that ComFLOW can well predict the physics of green water. The numerical methods adopted in ComFLOW have shown robustness to mesh refinement: when a fine enough grid is used, the results become grid independent. Also robustness with respect to green water impact on different cylinders is shown. The next step in the simulation of green water is to replace the dambreak starting situation with real waves.

The second topic of this paper has been the validation of ComFLOW using experiments of regular wave loading on a fixed SPAR buoy. The comparison between the calculated and measured forces is very satisfactory. Some problems were encountered at the outflow boundary where a local Sommerfeld condition has been used. Not enough water flew out of the domain which caused a raise of the fluid level. Also, peaks were observed in the pressure signal due to the numerical treatment of the outflow boundary. When the problems at the outflow boundary are solved, some attention needs to be given to the inflow boundary where reflections can occur caused by waves reflected by the SPAR buoy. Then, we will be able to simulate more periods of waves impacting on offshore structures.

ACKNOWLEDGMENT

The project is supported by the European Community under the FP5 GROWTH program; the authors are solely responsible for the present paper and it does not represent the opinion of the European Community. It is also supported by 26 parties from the industry (oil companies, shipyards, engineering companies, regulating bodies).

REFERENCES

- Botta, E.F.F., Ellenbroek, M.H.M., *A Modified SOR Method for the Poisson Equation in Unsteady Free-Surface Flow Calculations*, Journal of Computational Physics, **60**, 119-134, 1985.
- Buchner, B., *On the Impact of Green Water Loading on Ship and Offshore Unit Design*, PRADS '95, September 1995, Seoul.
- Daalen, E.F.G. van, Kleefsman, K.M.T., Gerrits, J., Luth, H.R., Veldman, A.E.P., *Anti-Roll Tank Simulation with a Volume of Fluid (VOF) Based Navier-Stokes Solver*, Symposium on Naval Hydrodynamics, Val de Rueil, September 2000.
- Gerrits, J., *Dynamics of Liquid-Filled Spacecraft*, PhD thesis, University of Groningen, December 2001.
- Gorf, P., Barltrop, N., Okan, B., Hodgson, T., Rainey, R., *FPSO Bow Damage in Steep Waves*, Proceedings Rogue Waves, Brest, November 2000.
- 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.
- Orlanski, I., *A Simple Boundary Condition for Unbounded Hyperbolic Flows*, Journal of Computational Physics **21**, 251-269, 1976.