Prediction of extreme wave loads in focused wave groups

Tim Bunnik[1], *Arthur Veldman*[2], *Peter Wellens*[3]
Marin,Wageningen[1]
University of Groningen[2]
Delft University of Technology[3]
The Netherlands

ABSTRACT

The Volume of Fluid (VoF) method is a promising tool to predict extreme wave loads on fixed and floating offshore structures. The VoF method described in this paper has been validated step by step by means of model tests like dam break flow, sloshing in LNG tanks and loads on fixed structures in extreme regular waves. Until recently, there was no means to generate a realistic extreme irregular wave in the VoF method. Traditionally, these are generated in time-domain simulations by picking extreme events from long-duration simulations that apply a random phase model to generate waves. Due to the fact that the computational times in the VoF method are large such an approach is not feasible. Instead, an approach based on wave focusing can be applied. The focused wave is designed by choosing the phases of the linear harmonic components such that they are identical at a certain target location. By means of linear dispersion, the wave time trace at the wave generator is predicted and subsequently the required motions of the wave generator. This approach was applied in model tests designed to validate the VoF model in extreme irregular waves. A simplified box-shaped structure was placed in a shallow water basin in the path of the focused wave group and the wave loads were measured. This paper focuses on the ability of the VoF method to reproduce these focused wave groups and the subsequent wave loads on a typical fixed structure, using the motions of the wave generator. The VoF method contains too much numerical dissipation to properly compute wave propagation over long distances (from the wave generator to the target location). Therefore, a non-linear potential flow method is used to simulate the wave propagation from the wave generator to the boundary of the computational domain of the VoF method (close to the target location). At the boundary of the VoF domain the wave kinematics from the potential flow method are used as boundary conditions for the VoF method. The VoF method is then used to determine the focused wave around the target location and the impact on a structure which is located there.

KEY WORDS: VoF; wave focusing; nonlinear waves; model tests; wave loads;

INTRODUCTION

The 2004/2005 hurricanes Ivan, Katrina and Rita in the Gulf of Mexico have led to renewed attention to extreme waves and their consequences for offshore structures as described by D.J. Wisch and E.G. Ward (2007) and G. Forristal (2007). This also involves prediction tools for possible extreme loads associated with these extreme waves. Model tests are commonly used for this purpose. Although model tests are the accepted standard and very valuable, they are expensive and do not show large insight in flow details during wave impact. Therefore a lot of effort is put into making CFD suitable for the prediction of extreme wave loads. The main problem at the moment is the large computational effort involved in CFD time-domain simulations Traditionally, model tests or simple time-domain simulations (based on linear diffraction wave loads) focus on long-duration (3 hours or more) sea states in which statistical information is gathered on extreme wave loads and distribution functions. Due to the large computational time, this is not possible in CFD and therefore it has to focus on a few isolated extreme events. There are several challenging questions that need to be answered:

1. What does a realistic extreme irregular wave look like?
2. When is an extreme wave extreme to the structure?
3. How is this extreme wave generated in CFD?

This paper addresses the third question and describes a method to generate long-crested extreme waves measured in the model basin in a Volume of Fluid method, and the subsequent loads on a structure in the path of the wave. The measurements include a regular wave and several focused wave groups. For all these waves, the motions of the wave generator have been measured and this signal is used to generate the waves in the simulations. The propagation of the waves from the piston-type wave maker towards the location of the structure is simulated by means of a non-linear potential flow code. The interaction with and the wave loads on the structure in the path of the wave are simulated by means of the VoF method. The wave elevation and wave orbital velocities obtained with the potential flow code are used as boundary conditions in the VoF method. This split up using two codes is done because of to the following reasons:

1. The VoF method contains considerable numerical dissipation and is not able to simulate the waves properly over the entire distance from the wave generator towards the structure (20 m).
2. The potential flow code is better able to simulate the propagation of the waves and is much less dissipative. However, this method is not able to simulate the breaking waves close to the structure and the interaction with the structure.
3. The VoF method is able to compute the interaction of the waves with the structure and the resulting wave loads.

This paper shows how to combine the two methods and shows a comparison of the combined methods with the results of model tests in focused wave groups.

The concept of wave focusing in model testing and simulations is not new and is for example applied in studies by Shuxue Liu et. al. (2005), Keyyong Hong et. al. (2004), Igor Ten and Hiroshi Tomita (2005), J. Zang et. al. (2007), J. Skourup and M. Sterndorff (2002) and G. Ducrozet et. al. (2006). These studies show that it is relatively easy to obtain high and steep waves using wave focusing (both directional and frequency focusing), but that advanced methods are needed to predict highly non-linear waves and the focus point correctly. Except for Zang et. al. (2007), these studies focus on the generation of the waves only and not on the wave-structure interaction. Zang et. al. (2007) focus on wave elevations near a vertical cylinder and obtain good results for not very steep waves (H/L=0.011). Wave loads are not considered. H. Bredmose et. al. (2006) show a comparison of their VoF method with results of model tests, including wave-structure interaction and wave loads. They reproduce the basin waves by means of linear theory in their VoF method, which has limitations for steeper waves.

THE NUMERICAL MODEL

The numerical model consists of two separate models which are interacting by means of a 1-way coupling:

1. HUBRIS, Finite Element Method (FEM) based on potential flow, capable of simulating non-linear wave propagation due to a moving wave generator, but excluding other fluid-structure interaction.
2. ComFLOW, Volume of Fluid method, capable of simulating non-linear free surface flows, including fluid structure interaction.

The results of the FEM are used to drive the VoF method by applying the FEM orbital velocities as boundary conditions in the VoF method. These methods are described in more detail in the following sections.

FINITE ELEMENT METHOD (HUBRIS)

HUBRIS is a non-linear time-domain simulation program for the simulation of long-crested wave propagation. Assuming the fluid to be incompressible and non-viscous with an irrotational velocity, and the free surface a single valued function of the horizontal coordinate, the following equations govern the fluid flow and free surface evolution:

$$\vec{v} = \nabla\Phi$$

$$\nabla \cdot \nabla\Phi = 0 \quad \text{in } \Omega$$

$$\nabla\Phi \cdot \vec{n} = V_n \quad \text{on } \Gamma$$

$$\frac{\partial\Phi}{\partial t} + \frac{1}{2}\left|\nabla\Phi\right|^2 + \frac{p_{air} - S'}{\rho} + g\eta = 0 \quad \text{on } z = \eta(\mathrm{x}, t)$$

$$\frac{\partial\eta}{\partial t} = \frac{\partial\Phi}{\partial z} - \frac{\partial\eta}{\partial x}\frac{\partial\Phi}{\partial x} - \frac{\partial\eta}{\partial y}\frac{\partial\Phi}{\partial y} \quad \text{on } z = \eta(\mathrm{x}, t)$$

Since the free surface is a single-valued function, overtopping (breaking) waves cannot be modeled (the method then breaks down). The fluid domain is divided into triangular finite elements. During the simulation, automatic re-gridding takes place if the fluid domain changes shape (near the free-surface and near the wave generator). The program computes the free surface, fluid velocities and pressures in four different, coupled domains:

1. Wavemaker domain, generating the waves
2. Grid domain
3. Dissipation domain, reducing high-frequency reflections
4. Sommerfeld domain for suppression of long waves

Each of these four domains are shortly discussed here

Wavemaker domain

The wave-maker domain is used to generate waves on the inflow boundary of the computational domain. Waves can be generated by means of an oscillating flap-type or piston-type wave generator. On the actual position of the wave generator, the water velocity normal to the wave generator is set equal to the normal velocity of the wave generator. The wave generator can have an arbitrary length, and an arbitrary hinge point. This way it is possible to simulate both piston type wave generators (limit of hinge point to infinity) and hinged wave flaps. Only the piston-type wave maker is considered in this paper.

Grid domain

The grid domain is next to the wave-maker domain. In this domain, the actual wave data of interest is computed and stored for further post-processing (wave elevation and velocity potential).

Dissipation domain

The dissipation damping domain is next to the grid domain and is used to damp the short wave components. This is done by means of exponential grid stretching. In the direction of wave propagation the size of the cells becomes larger by a factor α >=1. α =1 results in a uniform grid. This has a double effect:

1. The domain rapidly becomes very long thereby increasing the time it takes for reflection to return to the area of interest.
2. Due to the large cells, numerical dissipation becomes large, thereby dissipating most of the short wave components.

By means of a relatively small number of cells the reflection can be reduced to acceptable levels.

Sommerfeld domain

The Sommerfeld domain is a small domain next to the dissipation domain. This domain is used to damp the long wave components. At the outflow boundary of this domain, the Sommerfeld condition is applied:

$$\frac{\partial \varphi}{\partial t} + c_s \frac{\partial \varphi}{\partial x} = 0$$

With c_s the phase velocity of the wave component that should be absorbed. This condition, in theory, perfectly absorbs waves with wave phase velocity c_s. It can be shown that the reflection coefficient equals:

$$r = \frac{c_s - \omega / \kappa}{c_s + \omega / \kappa}$$

Where κ is the wave number. The phase velocity c_s is generally taken equal to the dominant transient long wave component (phase velocity $c_s = \sqrt{gh}$). Details of the proper choice of the determination of the phase velocity can be found in Westhuis (2001).

Output of the FEM are the wave elevation and velocity potential inside the fluid. By means of finite difference schemes, the wave orbital velocities are determined from the velocity potential and used as boundary condition in the VoF method.

VOLUME OF FLUID METHOD (COMFLOW)

ComFLOW is a non-linear time domain simulation program for the simulation of complex free surface flows including fluid structure interaction. Breaking waves can be modeled as well, but there is no physical model for their dissipation, and interaction with the air is not included yet (constant air pressure is applied).
The program discretises the Navier-Stokes equations on a fixed Cartesian grid. In a conservative form, they are given by:

$$\oint_{\partial V} u \cdot n dS = 0$$

$$\int_V \frac{\partial u}{\partial t} dV + \oint_{\partial V} u u^T \cdot n dS =$$

$$-\frac{1}{\rho} \oint_{\partial V} (pn - \mu \nabla u \cdot n) dS + \int_V F dV$$

Here, ∂V is the boundary of volume *V*, *u* = (*u*,*v*,*w*) is the velocity vector in the three coordinate directions, *n* is the normal of volume *V*, ρ denotes the density and *p* is the pressure. μ denotes the dynamic viscosity and *F* = (*Fx*,*Fy*,*Fz*) is an external body force, for example gravity.

The variables are staggered, which means that the velocities are defined at cell faces, whereas the pressure is defined in cell centers. The body geometry is piecewise linear and cuts through the fixed rectangular grid. Volume apertures (*Fb*) and edge apertures (*Ax*, *Ay*, and *Az*) are used to indicate for each cell which part of the cell and cell face respectively is open for fluid and which part is blocked by solid geometry. To track the free surface, the Volume-of-Fluid function *Fs* is used, which defines the fraction of the cell that is filled with fluid (so *Fs* = 0 if no fluid is present in the cell, *Fs* =1 if the cell is completely filled with fluid and *Fs* is between 0 and 1 if the cell is partly filled with fluid). The Navier-Stokes equations are applied in every cell containing fluid. Cell labeling is introduced to distinguish between cells of different characters. First the cells which are completely blocked by solid geometry are labeled as B(oundary) cells. These cells have volume aperture *Fb*=0. Then the cells which are empty, but are open to fluid flow are labeled E(mpty). The adjacent cells, containing fluid, are labeled S(urface) cells. The remaining cells are labeled F(luid) cells. Figure 1 shows an example of the labeling in case of a wedge that penetrates a fluid.

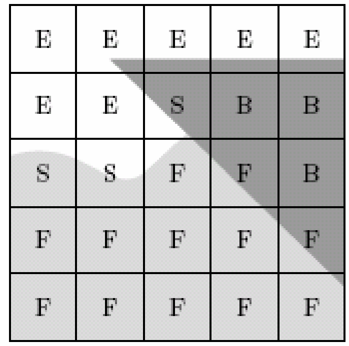

Figure 1: Example of cell labeling in the VoF method in case of a wedge penetrating a fluid.

The Navier-Stokes equations are applied in every cell containing fluid. The air is modeled as a void, having a constant pressure. The VoF method can deal with overturning flows, but air entrapment is not modeled. In a separate research project, the VoF method is being extended to include a 2nd, compressible air phase as described by R. Wemmenhove, E. Loots, R. Luppes and A.E.P. Veldman (2005).

It is possible to place arbitrary fixed or moving structures in the flow. The geometry of the structure is defined by basic elements like bricks, wedges, spheres and cylinders. This makes it possible to simulate the wave loads on arbitrary offshore structures. In case of symmetrical flow (a symmetrical construction in a long-crested head wave) it is possible to apply a symmetry plane. This saves considerable computational time.

The method uses first-order upwind difference schemes for the convective terms in the Navier-Stokes equations and a first-order free-surface displacement algorithm. This proves to be very stable, but also results in significant dissipation in the waves. Therefore, the method can only be applied in small domains and incident waves need to be generated by means of other theories or (less-dissipative) simulation tools.

Pressure damping can be applied on the free surface behind the structure to damp outgoing waves and acts as a numerical beach. At the free surface, additional damping is applied according to

$$p_{damp}(t, x, \varsigma) = \alpha(x) w(t, x, \varsigma)$$

Where p_{damp} is an additional damping applied on the free surface, w is the vertical water velocity on the free surface and $\alpha(x)$ a damping function which can be tuned to absorb the waves at good as possible. The damping function applied in the simulations shown in this paper is a linearly increasing function, starting just behind the structure with a slope of 0.1 Ns/m^4. Besides this beach, grid stretching is used to obtain a long domain to further reduce the amount of wave reflection.

In the area between the inflow and the structure pressure damping and stretching cannot be applied since this would affect the incoming waves as well. This means that diffracted waves which are propagating back from the structure to the inflow boundary are fully reflected. This can be a serious problem in case of full bodies with heavy diffraction. At the moment a special boundary condition is being developed

(Generating and Absorbing Boundary Condition; GABC) that can deal with this problem.

The boundary conditions in the VoF method have been implemented such that the user has the possibility to specify fluid velocities and the wave elevation at the boundaries. This makes it possible to use results from other (less dissipating) wave generation methods as input to the VoF method. The VoF method can then deal with the complex interaction with the structure in a small domain.

MODEL TESTS

A special series of model tests was carried out to validate the suggested approach. A rectangular structure was placed in the model test basin at a distance of 20 m from the wave generator. A captive setup was used in which the structure was fixed in a force frame which could measure the forces and moments on the structure due to the interaction with the waves. The depth of the basin was 1 m. Figure 2 shows the setup of the tests. Table 1 shows the dimensions of the block. Several photos of the test are shown in Figures 3 and 4.

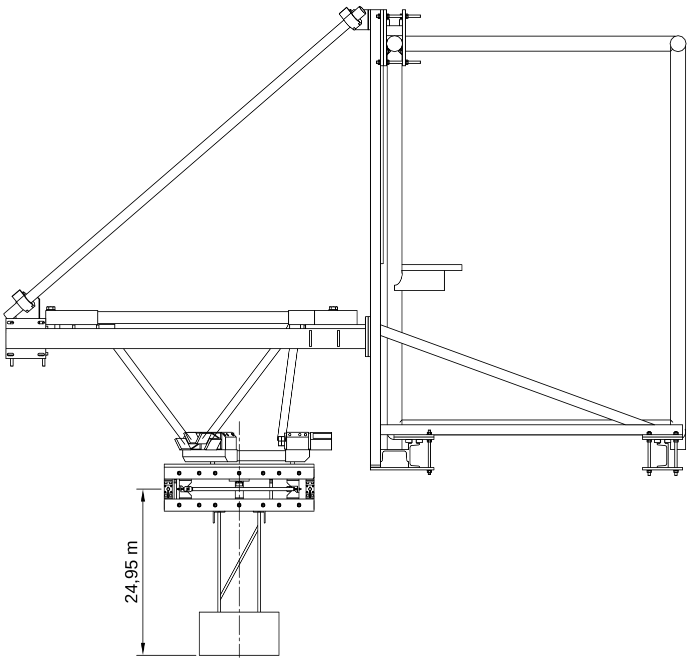

Figure 2: Captive setup for force measurement on block.

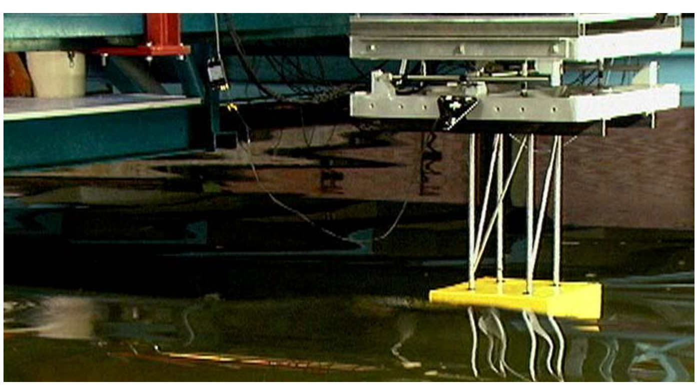

Figure 3: Captive block in regular wave test.

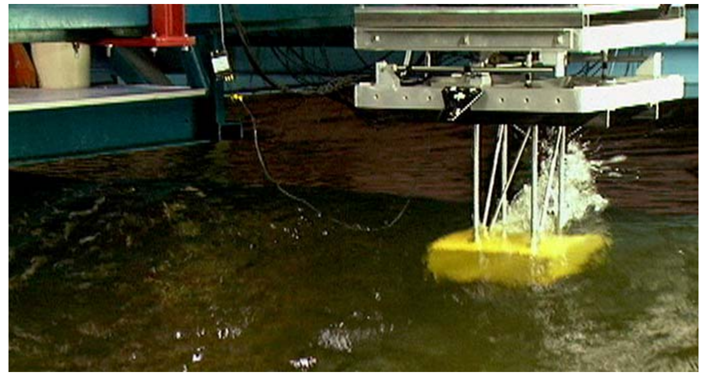

Figure 4: Captive block in focused wave test.

Description	Unit	Value
Length	m	0.400
Width	m	0.400
Height	m	0.212
Draft	m	0.155

Table 1: Main particulars of fixed block

RESULTS

Wave calibration

Prior to the simulations with the box, simulations without the box were carried out to compare the undisturbed simulated wave elevation with the measured wave elevation. Since the considered waves are long-crested, these simulations could be done in two dimensions (1 grid-cell in the transverse direction). The domain in the FEM method stretched from the wave generator (X=0) to X=155 m. The box is later to be placed at X=20 m from the wave generator. The domain in the VoF method stretched from X=18 m to X=52.3 m. The duration of the VoF simulations was 15 s for the regular wave (period 2.92 s, crest height 0.27 m) and 20 s for the focused wave group. At the boundary of the VoF domain (X=18 m) the wave orbital velocities were computed from the FEM results and used as boundary condition. This was done only for the focused wave group. For the (steeper) regular wave, the FEM broke down after some time and the wave orbital velocities were therefore computed by the method of M.M. Rienecker and J.D. Fenton (1981) instead.

Figure 5 shows the undisturbed wave elevation for the regular wave. The following can be noticed:

- Up to the time of break down, a good agreement between the FEM and the measurements is obtained, including the transient effect, since the piston motions in the model tests and the FEM simulation were the same.
- The transient effect in the VoF method is different from the model tests since the VoF wave is generated with the method of Rienecker and Fenton (slowly starting up the velocities with a smooth ramp function) and not with a moving piston.
- After the start-up the solution of the VoF method is in very good agreement with the measurements in the time frame between 10 and 13 seconds. This timeframe can therefore be used for comparison with the wave loads.
- There is hardly any difference between the VoF results on the 2 different grids.

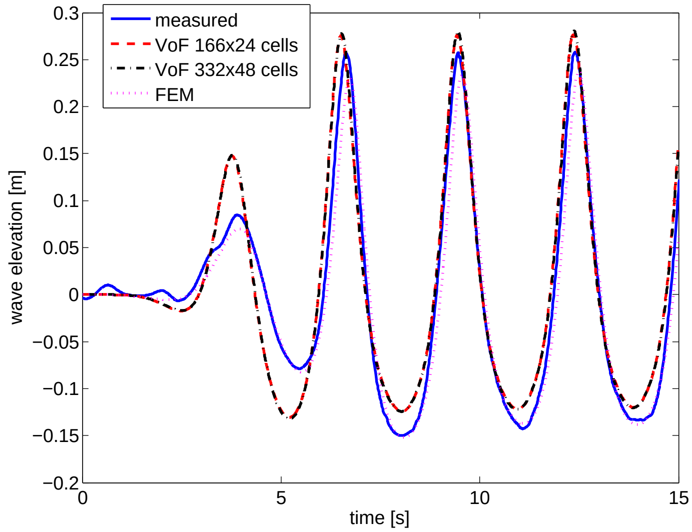

Figure 5: Measured and simulated regular wave with period 2.92 s.

Figure 6 shows the undisturbed wave elevation for the focused wave group. The following can be noticed:

- There is a reasonable agreement between the FEM and the measurements. The FEM is not able to predict the high-frequency oscillations around T=12-13 s. Furthermore, the through is not deep enough and the crest somewhat too high.
- Since the FEM results are input to the VoF model, similar differences between the measurements and the VoF model are found. The crest height is somewhat better predicted because the dissipation in the VoF model lowers the FEM crest a little.

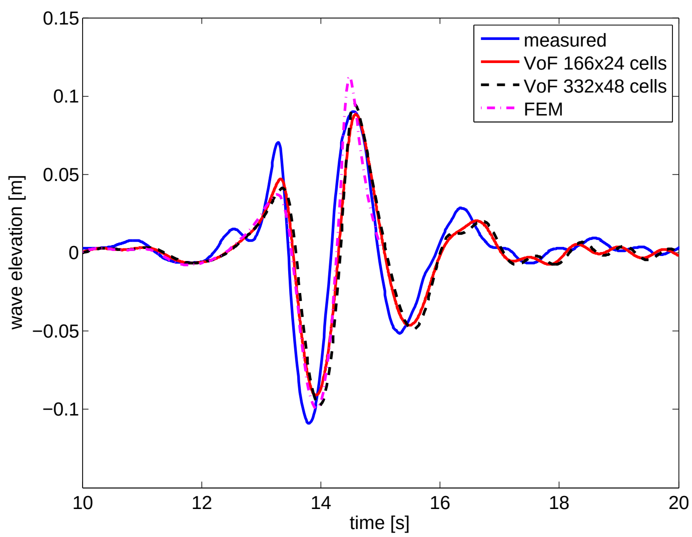

Figure 6: Measured and simulated focused wave group.

Wave loads

After comparing the undisturbed waves, the simulations were repeated with:

- The domain extended in the transverse direction (1.63 m to each side of the structure).
- The box placed in the path of the wave at X=20 m.

Figure 7 shows 2 snapshots of the simulation in regular waves at the moment the wave crest is submerging the structure.

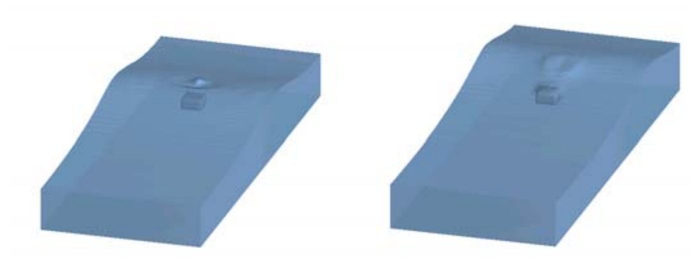

Figure 7: Snapshots of regular wave submerging the block structure at 6.5 (left) and 6.9 (right) seconds.

Figures 8 and 9 show the horizontal and vertical wave loads on the block structure in the regular wave. The following can be noticed:

- There is a considerable difference between the VoF results for the different grids. The horizontal peak loads are higher for the coarsest grid and the secondary peak in the vertical load is underestimated for the coarse grid. When refining the grid, the forces are getting close to the measurements. This is probably related to the fact that the VoF method does not use pressure interpolation and extrapolation, but uses the pressure closest by to compute the pressure loads on the structure. It can be concluded that a grid-independent solution has probably not yet been reached. However, due to the large computational times (4 days on a single processor for the finest grid) it was not found practical to double the number of grid cells once more.
- The general agreement between the measured and computed loads on the refined grid is good. The secondary peak and the height of the primary peak in the vertical load are predicted well. The secondary peak is caused by a small breaking wave on top of the wave crest, collapsing on the surrounding fluid when the structure is submerged (see Figure 7). The duration of the zero horizontal force (corresponding to the time the wave through is below the structure and the structure is completely dry) is predicted very well.

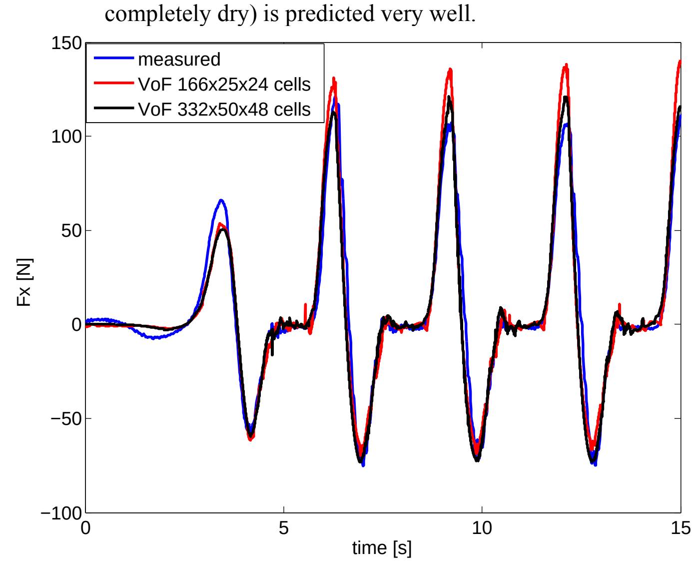

Figure 8: Measured and simulated horizontal wave loads on block structure in regular wave.

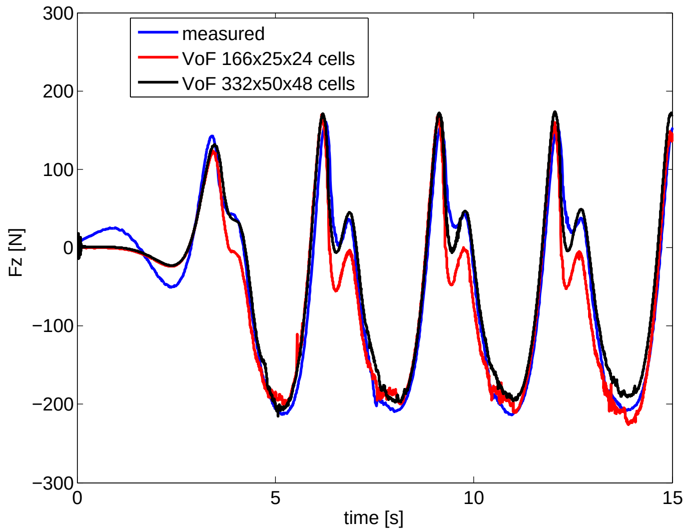

Figure 9: Measured and simulated vertical wave loads in regular wave.

Figures 10 and 11 show the horizontal and vertical wave loads on the block structure in the focused wave group. The following can be noticed:

- There are differences between the measured and simulated wave loads, but these seem to be in the order of the mismatch between the measured and simulated undisturbed wave by the FEM. A better prediction of the undisturbed wave would most likely have resulted in a better prediction of the wave loads. Considering the difference between the undisturbed waves, the agreement in the wave loads seems to be predicted well.
- There is a high-frequency vibration in the VoF result on the coarse grid. The reason for this is unclear.

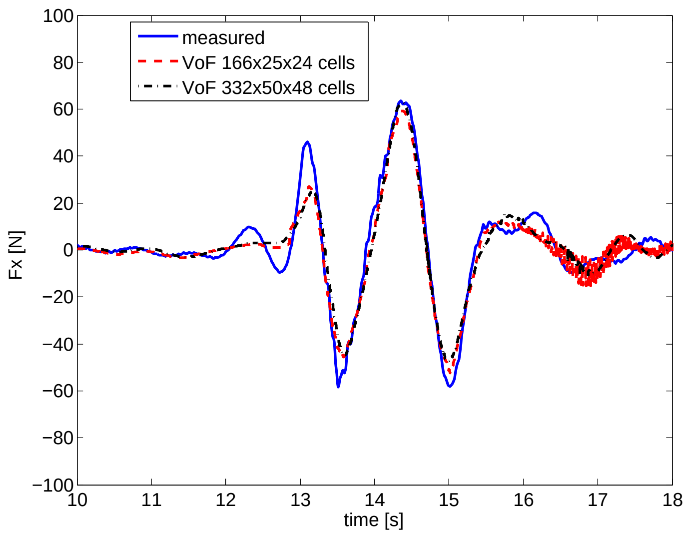

Figure 10: Measured and simulated horizontal wave loads in focused wave group.

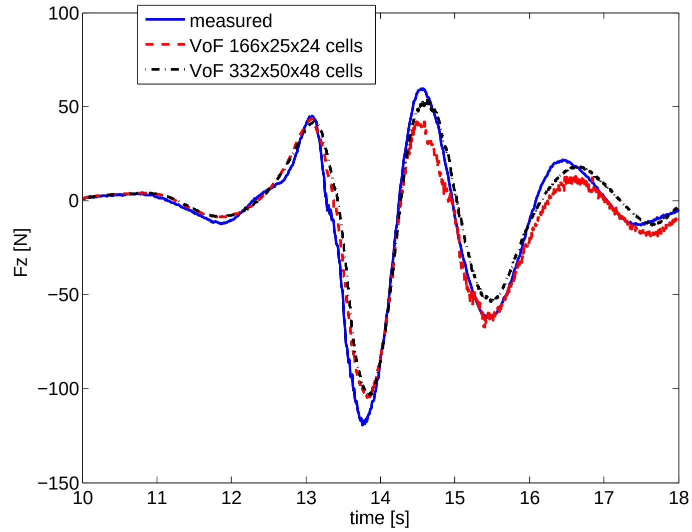

Figure 11: Measured and simulated vertical wave loads in focused wave group.

Besides the focused wave group shown in this paper, higher and steeper focused wave groups were tested as well. The generation of these wave groups leads to instabilities in the FEM and therefore these could not be simulated with the VoF method. At the moment, alternative methods are being considered for the generation of these steep wave groups. E. van Groezen et. al. have recently developed a new method for accurate simulation of uni-directional surface waves. This method is at the moment being made suitable for coupling to the VoF method.

CONCLUDING REMARKS

A VoF method was presented to compute non-linear wave loads on offshore structures. By coupling the VoF method to a wave generation program (FEM based on potential flow), focused wave groups measured in the model test basin can be simulated provided that they are not too steep. The VoF method is restricted to a small domain near the structure exposed to the waves. This prevents the waves to dissipate in the VoF method when propagating from the wave generator to the structure. The wave orbital velocities computed by the (far less dissipating) FEM method are applied as boundary condition in the VoF method. A comparison with model tests shows that the wave loads are predicted well.

The remaining issues are the generation of very steep waves and the computational time in the VoF method. To simulate steep waves, a coupling is now being made to a more robust and accurate wave generation method described by E. van Groezen et. al. To speed-up the computational process, the VoF method will be extended with parallel computing and local grid refinement. Furthermore, by improving the absorbing boundary conditions, it will be made possible to use a smaller computational domain.

REFERENCES

Giorgio Contento , Riccardo Codiglia and Fabrizio D'Este (2001). "Non-linear effects in 2D transient nonbreaking waves in a closed flume," *Applied Ocean Research*, Vol 23, No 1, pp 3-13.

Jaap-Harm Westhuis (2001). "The numerical simulation of nonlinear waves in a hydrodynamic model test basin," PhD thesis, University of Twente.

Shuxue Liu et. al. (2005). "Experimental study on multi-directional wave focusing and breaking in shallow water", *Proceedings 15th ISOPE conference,* Seoul, pages 486-492.

Keyyong Hong et. al. (2004). "Energy dissipation in breaking waves generated by directional and multi-frequency focusing in deep water", *Proceedings 14th ISOPE conference,* Toulon, pages 299-305.

Igor Ten and Hiroshi Tomita (2005). "Creation and annihilation of extremely steep transient wave", *Proceedings 15th ISOPE conference,* Seoul, pages 10-17

J. Zang et. al. (2007). . "A new Boussinesq model for wave run-up on curved structures using Cartesian cut-cell grids", *Proceedings 17th ISOPE conference,* Lisbon, pages 2084-2091.

J. Skourup and M. Sterndorff (2002). "Deterministic reproduction of nonlinear waves", *Proceedings 21st OMAE conference*, Oslo, Norway.

H. Bredmose et. al. (2006). "Numerical reproduction of extreme wave loads on a gravity wind turbine foundation", *Proceedings 25st OMAE conference*, Hamburg, Germany.

G. Ducrozet et. al. (2006) "Implementation and validation of nonlinear wave maker models in a HOS numerical wave tank", *Proceedings 16th ISOPE conference,* San Francisco, pages 334-341.

D.J. Wisch and E.G. Ward (2007). "Offshore standards – the impact of hurricanes Ivan/Katrina/Rita". *Proceedings 26st OMAE conference*, San Diego, USA.

G. Forristal (2007). "Wave crest height and deck damage in hurricanes Ivan, Katrina and Rita", *Proceedings OTC*, Houston, USA.

E. van Groezen et. al. "Accurate modelling of uni-directional surface waves". Submitted for publication in Elsevier.

E.F.G. van Daalen, J. Gerrits, G.E. Loots and A.E.P. Veldman (2000). "Free surface anti-roll tank simulations with a volume of fluid based Navier-Stokes solver". In: T. Miloh and G. Zilman (eds.) *Proc. 15th Int. Workshop on Water Waves and Floating Bodies,* Caesarea, Israel, Feb 27 - Mar 1, 2000.

E. Loots and B. Buchner (2004). "Wave run up as important hydrodynamic issue for gravity based structures". In: *Proc. 23rd Int. Conf. Offshore Mech. Arctic Engng,* Vancouver, paper OMAE2004-51084 (2004).

T. Bunnik and B. Buchner (2004). "Numerical prediction of wave loads on subsea structures in the splash zone". In: J.S. Chung, K. Izumiyama, M. Sayed and S.W. Hong (eds.) *Proc. 14th Int. Offshore and Polar Engineering Conference ISOPE2004,* Vol. I, pp. 284-290 (2004).

K.M.T. Kleefsman, G. Fekken, A.E.P. Veldman, B. Iwanowski and B. Buchner (2005). "A Volume-of-Fluid based simulation method for wave impact problems". *J. Comp. Phys.* 206 (2005) 363-393.

R. Wemmenhove, E. Loots, R. Luppes and A.E.P. Veldman (2005). "Modelling two-phase flow with offshore applications". In: *Proc. 24th Int. Conf. Offshore Mech. Arctic Eng.,* Halkidiki, Greece (2005) paper OMAE2005-67460.

T.B. Johannessen, S. Haver, T. Bunnik and B. Buchner (2006). "Extreme wave effects on deep water TLP's - Lessons learned from the Snorre A model tests". In: *Proc. Deep Offshore Technology 2006*, Houston, Nov. 28-30 (2006).

R. Wemmenhove, R. Luppes, A.E.P. Veldman and T. Bunnik (2007): "Numerical simulation of sloshing in LNG tanks with a compressible two-phase model". In: *Proc. 26th Int. Conf. Offshore Mech. Arctic Eng.,* San Diego (2007) paper OMAE2007-29294.

M.M. Rienecker and J.D. Fenton (1981): "A Fourier approximation method for steady waves", In: *Journal of Fluid Mechanics,* Vol. 104 (1981), pp. 119-137.