

Simulation of focused waves and impacts in a numerical wave basin

T.H.J. Bunnik⁽¹⁾ and E.W.C. van Groesen⁽²⁾

⁽¹⁾*MARIN, Wageningen, The Netherlands*

⁽²⁾*University of Twente, Enschede, The Netherlands*

ABSTRACT: The Volume of Fluid (VoF) method is promising in predicting 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 means of long-duration simulations that apply a random phase model to generate waves. The occurrence of extreme waves is than a matter of patience. 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 as boundary condition. 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.

1 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 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 several 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. Ducroz 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.

2 THE NUMERICAL MODEL

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

1. Non-linear wave model based on potential flow and the AB equations to describe the free surface waves, capable of simulating non-linear wave propagation, but excluding 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 non-linear wave model are used to drive the VoF method by applying the wave orbital velocities as boundary conditions in the VoF method. These methods are described in more detail in the following sections.

3 NON-LINEAR POTENTIAL FLOW WAVE MODEL

The non-linear wave model is based on the variational principle for inviscid irrotational water flow. Zakharov's Hamiltonian formulation is used to obtain a description in surface variables only. Keeping exact dispersion properties of infinitesimal waves, the kinetic energy is approximated. Invoking a uni-directionalization constraint leads subsequently to the AB-equation, a KdV-type of equation that is also valid on infinitely deep water, that is exact in dispersion for infinitesimal waves, and that is second order accurate in the waveheight. The AB equations itself can be found in E. van Groesen et. al. (2008).

The result of the AB-equations is the non-linear fluid potential at the still water level. In order to couple the results of the AB equation with the VoF method, the fluid velocity between the seabed and the actual free surface is needed. In order to derive the wave orbital velocities between the seabed and the still water level, a pseudo-analytical approach is used:

$$u(x, z, t) \hat{=} \sqrt{\frac{2gk}{\sinh 2kD}} \cosh k(z + D) \hat{\eta}(k, t) \quad (1)$$

$$w(x, z, t) \hat{=} -i\hat{u}(k, z, t) \tanh k(z + D)$$

Where $\hat{\eta}$ and \hat{u} are the space Fourier Transforms of the surface elevation and the horizontal velocity. The velocities above the still water level are obtained using a linear extrapolation as follows:

$$u = U_0 + z \partial_x W_0(x) \quad (2)$$

$$w = W_0$$

Where U_0 and W_0 are the horizontal and vertical fluid velocities at the still water surface.

4 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:

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

$$\int_V \frac{\partial u}{\partial t} dV + \oint_{\partial V} uu^T \cdot n dS = \quad (3)$$

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

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.

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. Luppés 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

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

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 as 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/m⁴. 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.

4 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 1 shows the setup of the tests. Table 1 shows the dimensions of the block.

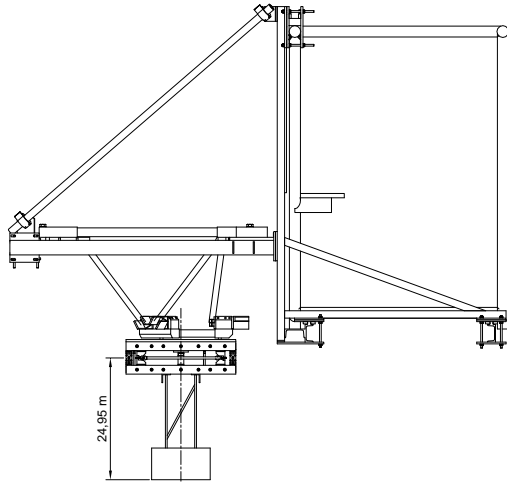


Figure 1: Captive setup for force measurement on block (left) and the situation in the basin (right).

Table 1: Main particulars of fixed block

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

5 RESULTS

5.1 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 for the AB equation stretched from the wave generator ($X=0$) to some distance behind the position of the box (20 m from the wave generator), applying a constant grid size of 0.04 m (which was found sufficient for convergence). Of course the box was not modeled for the undisturbed waves. The boundary condition for the AB equations is the free surface elevation at the first (upstream) node. Since the wave elevation was not measured near the wave maker, this wave elevation was estimated using the measured wave maker motion and its linear transfer function. Two focused wave groups are considered in this paper, the first with a crest height of 8 cm (low focused wave) and the second with a crest height of 23 cm (high focused wave).

The domain in the VoF method stretched from $X=18$ m to $X=52.3$ m. The duration of the VoF simulations was 20 s for both focused wave groups. At the boundary of the VoF domain ($X=18$ m) the wave orbital velocities were computed from the AB results and used as boundary condition.

5.1.1 Low focused wave

Figure 3 shows the results for the (low) focused wave. On the left, a comparison between the measured and AB-simulated undisturbed wave elevation is shown. It can be noticed that the agreement between the results from the AB equations and the measurements is very good. In the middle, a comparison between the VoF-simulated undisturbed wave elevation (three different grid sizes) using the AB results as input is shown. The

differences between the last two grids are again small. On the right, a comparison between the VoF-simulated undisturbed wave elevation (finest grid) and the measurement is shown. The differences with the measurements are somewhat larger compared with the AB results (Figure 3, left). This is due to the fact that the VoF method has difficulties modeling the shortest wave components in the focused wave group due to numerical dissipation.

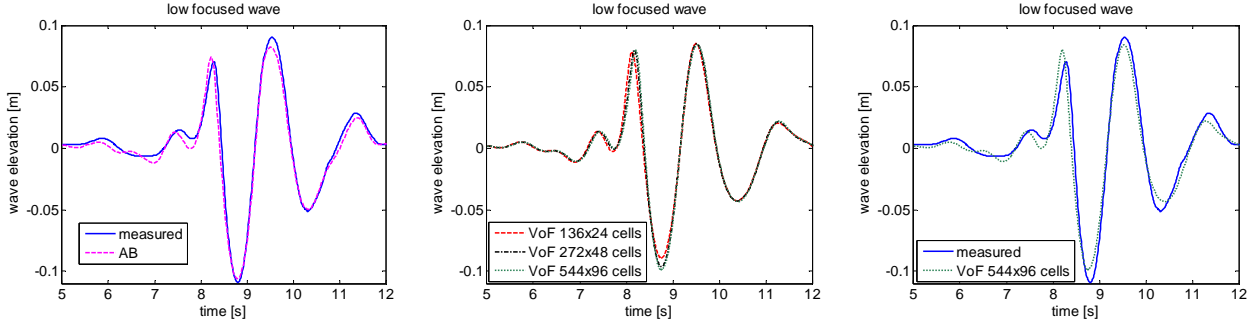


Figure 3: Measured and simulated, low focused wave. Measured versus AB (left), VoF results for three grid sizes (middle) and measured versus VoF (right).

5.1.2 High focused wave

Figure 4 shows the results for the (high) focused wave. On the left, a comparison between the measured and AB-simulated undisturbed wave elevation is shown. It can be noticed that the agreement between the results from the AB equations and the measurements is good. Differences can be explained due to the fact that this wave was near breaking at this location (20 m from the wave generator) and was actually seen to be breaking just behind the test setup. In the middle a comparison between the VoF-simulated undisturbed wave elevation (three different grid sizes) using the AB results as input is shown. It can be seen that the wave collapses in the VOF simulation on the 2 finest grids. Apparently the numerical breaking limit is reached somewhat earlier than the physical breaking limit. On the right, a comparison between the VoF-simulated undisturbed wave elevation (finest grid) and the measurement is shown. Since the high crest has collapsed, the comparison is poor in this point of time, but quite good otherwise.

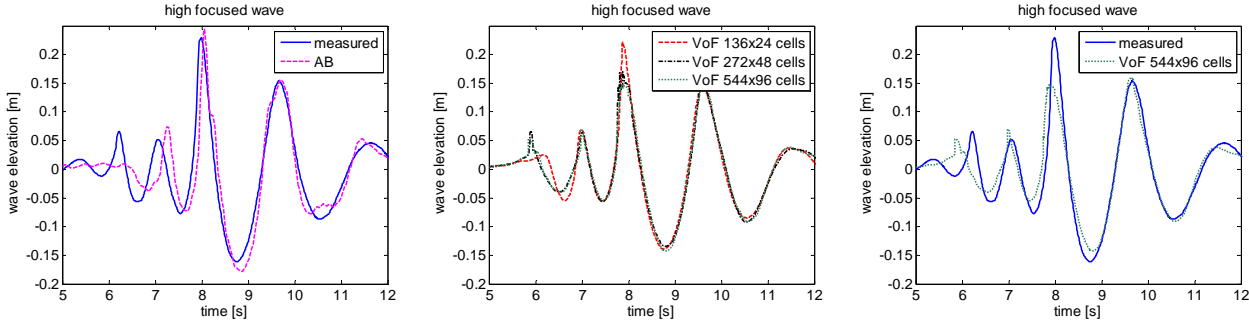


Figure 4: Measured and simulated, high focused wave. Measured versus AB (left), VoF results for three grid sizes (middle) and measured versus VoF (right).

5.2 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 5 shows the horizontal and vertical wave loads on the block structure in the focused wave group with maximum crest height 8 cm. The following can be noticed:

- There are differences between the measured and simulated vertical wave loads, especially in the deepest wave trough. These are larger than what would be expected from the differences in the undisturbed waves (which were small). Since there is no water running over the block, these loads are only caused by pressures at the bottom of the block. Apparently these are under predicted. The horizontal loads seem to be predicted very well.
- There is a high-frequency vibration in the VoF result on the finest grid. The reason for this is unclear. The high-frequency vibration in the measured horizontal loads is the resonance frequency of the captive setup.

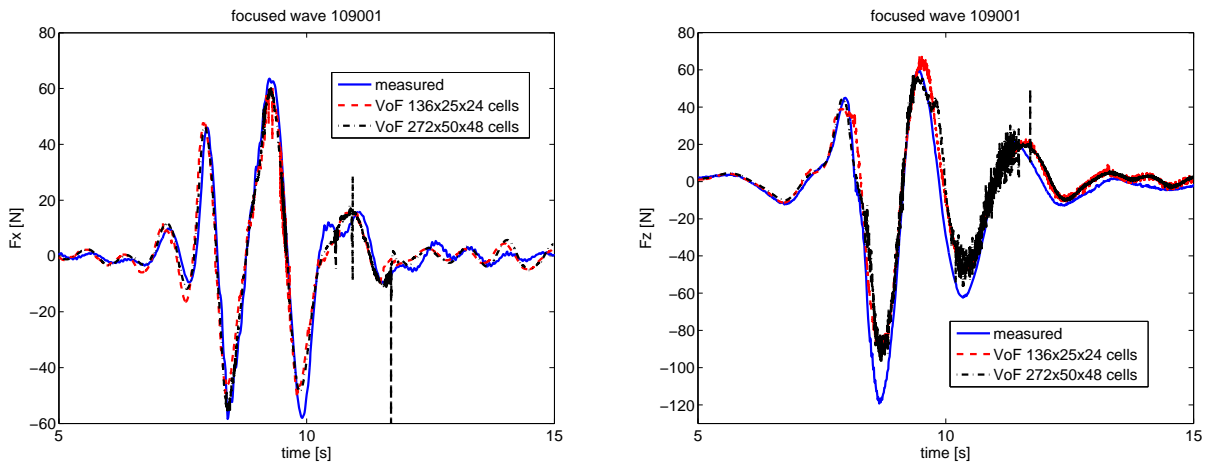


Figure 5: Measured and simulated horizontal wave loads in focused wave group with maximum crest height 8 cm.

Figure 6 shows the horizontal and vertical wave loads on the block structure in the focused wave group with maximum crest height 23 cm. The following can be noticed:

- Again, there are differences between the measured and simulated vertical wave loads and a better agreement in the horizontal loads.
- Grid refinement has a considerable effect on the vertical loads and gets the results closer to the measurements. Possibly an even finer grid has to be used to obtain converged results, but this was practically not possible due to the large computational times.

6 CONCLUDING REMARKS

A VoF method was presented to compute non-linear wave loads on offshore structures. By coupling the VoF method to a non-linear wave generation program based on potential flow and the recently derived AB-equations, focused wave groups measured in the model test basin can be simulated. 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) potential flow method are applied as boundary condition in the VoF method. A comparison with model tests shows that the wave loads are predicted quite well.

The main issue with the VoF model is the excessive computational time. 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.

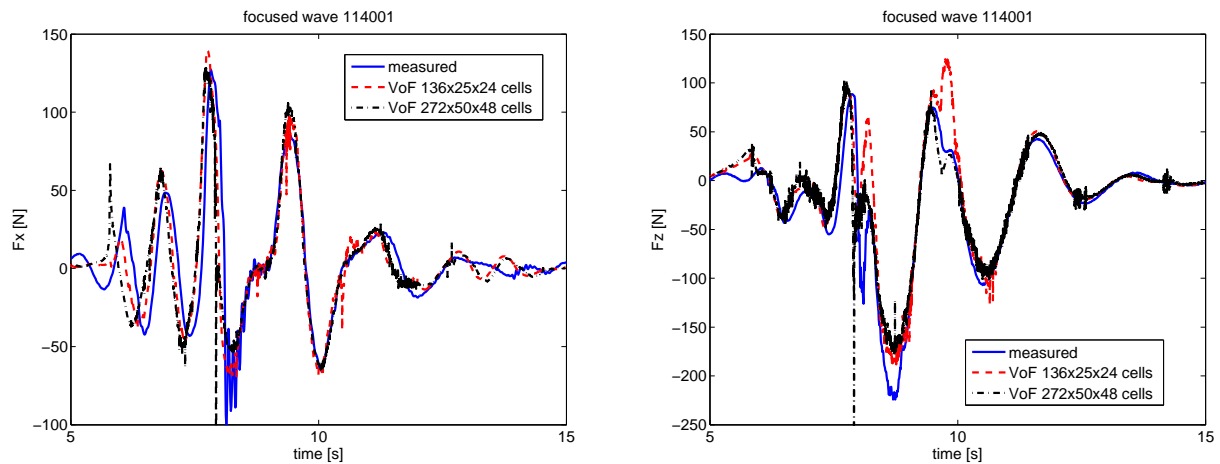


Figure 6: Measured and simulated horizontal wave loads in focused wave group with maximum crest height 23 cm.

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. Ducroz 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. Luppés 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. Luppés, 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.
- E. van Groesen et al. "Accurate modelling of uni-directional surface waves". *Journal of Computational and Applied Mathematics* (to be published).