Focused waves and wave-structure interaction in a numerical wave tank

J. Westphalen*[,a], D.M. Greaves[b], C.J.K. Williams[c], A.C. Hunt-Raby[b], J. Zang[c]

[a] *Center for Ocean Energy Research, Electronic Engineering, National University of Ireland, Maynooth, Co. Kildare, Ireland*

[b] *School of Marine Science and Engineering, University of Plymouth, Drake Circus, Plymouth, PL4 8AA, UK*

[c] *Department of Architecture and Civil Engineering, University of Bath, Bath, BA2 7AY, UK*

Abstract

Sustainable and efficient design solutions are the aim for any engineer. In offshore engineering forces resulting from extreme wave impact are of special interest as these challenge the structure and the crew working in this harsh environment. Theoretical models tend to be limited to linear or weakly non-linear situations and are unable to predict the violent and turbulent effects of breaking waves in combination with wave run up on structures or green water loading. The classic approach for such cases is to carry out scale model tests in a physical wave tank and measure the forces, water levels and flow velocities at some chosen locations.

In this paper another approach is investigated, that uses fully non-linear Computational Fluid Dynamics calculations, and has the potential to investigate the design in different conditions at full scale. The suitability of the

*Corresponding author

Email address: jan.westphalen@eeng.nuim.ie (J. Westphalen)

use of the numerical techniques implemented in commercial CFD packages for design of offshore structures under extreme waves is considered. The numerical schemes are used to simulate wave focussing to generate an extreme wave in a numerical wave tank and for regular wave interaction with structures. Non-linear effects of extreme wave interaction are demonstrated and the implications for a numerical wave tank are discussed. Also the forces on horizontal and vertical cylinders, which represent simple models of offshore structures, are calculated. The predicted results generally compare well with physical experiment data, both in wave surface profile prediction and in wave forces on structures, and conclusions are drawn regarding the suitability of the numerical approaches for these applications.
Key words: focused waves, NewWave, horizontal cylinder, vertical cylinder, Computational Fluid Dynamics, Control-Volume Finite Element method, Finite Volume method

---

Wave interaction with structures is a large and important research area. Due to the complexity of the wave breaking process and the interaction of a wave with a structure due to wave run-up and green water effects, a great deal of effort is required to investigate the physics.

One of the most common approaches to model testing in extreme waves is by physical tank testing. The big advantage of this method is that the real hydrodynamics are modelled and can be measured, filmed and reproduced as often as necessary, although physical testing is limited by scale restrictions. Also, with a physical model it is fairly quick to carry out a large number of tests. However, to obtain representative data sets this might well be necessary, as only a certain number of wave gauges, Acoustic Doppler Velocimeters or pressure sensors can be placed in the flow without affecting it. Scaling effects from model to full scale might occur as well. To overcome this, analytical methods have been developed. Empirical approaches, such as the Morison formula (1) to estimate the forces on a pile, can be used. As empirical methods depend on flow coefficients that have to be estimated from measurements, they are often not generally valid for all flow regimes. In the case of the Morison formula forces are only estimated correctly when applied to compact bodies, (Sarpkaya and Issacson , 1981) and it does not provide complete pressure distributions on these bodies.

Potential flow methods may be used in offshore engineering to compute the fluid flow in a more realistic way. A general overview of the different methods is given by Newman and Lee (2002). Here the model is discretised by boundary elements or panels. For each panel the pressures and the flow velocity potential is calculated. The potential flow formulation is obtained by simplifying the Navier-Stokes equations assuming non-viscous and non-rotational incompressible fluid flow. The velocity derived equation set is simpler and faster to solve than the full Navier-Stokes equations. This makes it efficient for linear and weakly non-linear wave structure interaction problems, although the methods are usually restricted to non-breaking waves and wave problems up to the breaking point, as described by Zang *et al.* (2006). More recent advances have been reported by Ma and Yan (2009) and Yan and Ma (2010), who have developed a combined Eulerian-Lagrangian technique for 3-D wave breaking. Although node adjustment at the free surface can be time consuming, they show computational speeds of at least 10 times the fastest of the BEM methods (Yan and Ma , 2008).

For the high non-linear phenomenons of wave breaking, green water and possible violent body motion Computational Fluid Dynamics(CFD) can be used. Eulerian methods use a mesh to represent the geometry. On this mesh the equations describing the fluid flow, such as the Euler and Navier-Stokes equations, are solved for each mesh element. Lagrangian methods, such as Smooth Particle Hydrodynamics (SPH), model the interaction between particles that represent the fluid rather than using a mesh. The advantage of CFD is that in principle it is valid for all flow regimes in offshore engineering. Hence it can be applied to all the problems described earlier, but also for overturning flows and where viscous effects are important. Furthermore the simulations do not need to be scaled as is necessary for physical tank tests, although they are often validated against such results due to a lack of full scale data available. However, the computational effort for CFD is high and the accuracy of the prediction not assured compared to the empirical and potential flow methods. With the development of computational power and improvements in the efficiency of the numerical simulation models, engineers now have the possibility of backing up their measurements or extending their measured data with calculated results from validated CFD simulations. For this work two different Computational Fluid Dynamics (CFD) codes are applied to classic offshore engineering applications, the generation of extreme waves and the interaction of regular waves with cylinders.

Section 1 describes the governing equations of the fluid flow and the two solvers, a Finite Volume approach (Section 1.2) and a control-volume Finite element method (Section 1.3), by which these equations are solved.

The first set of tests is described in Section 2. They encompass the generation of extreme waves where several relatively small waves are superposed to form one focused wave at a specified location in the tank. The principles of this technique are described in Baldock et al. (1996). This paper deals with the numerical setup for the generation of three NewWave cases as published in Ning *et al.* (2009). The test cases increase in the level of non-linearity up to the point where the waves almost break. Comparisons of surface elevation predictions to high quality experimental data are made using input signals defined by linear wave theory and linear plus 2^{nd} order.

The interaction between waves and structures is conducted using fixed vertical and horizontal cylinders. Firstly, semi-submerged horizontal cylinders in regular waves are simulated. The topic itself has been investigated extensively and many experiments by a number of authors are described in the literature. Hogben *et al.* (1977) review many of these early studies.

Due to developments in offshore exploration techniques for oil and gas it became important to know the stresses on oil rigs and drilling platforms. Morison *et al.* (1950) investigated the forces on piles systematically and proposed a simple formula which became known as the Morison equation

$$F = \frac{1}{2}\rho C_D D u \left| u \right| + \frac{1}{4}\,\rho \pi D^2 C_M \frac{du}{dt}. \tag{1}$$

Here, F is the horizontal force per unit length on the cylinder of diameter D and u is the horizontal component of the water particle velocity. C_D and C_M are the coefficients for drag and inertia respectively. The range of values for C_D and C_M, used with Morison's equation are outlined by Hogben *et al.* (1977), a handy tool to estimate the forces per unit length for a pile. However, Equation (1) was developed from tank tests driven with small sinusoidal waves, which makes it applicable for specific cases as stated in Keulegan and Carpenter (1958). Keulegan and Carpenter's objective was to extend the Morison equation (1) by a supplementary function ΔR to represent the forces more truly when C_D and C_M were considered to be constant throughout the whole wave cycle. Also they introduced a period parameter, which later became the Keulegan-Carpenter number N_{KC}, as

$$N_{KC} = \frac{AT}{D}, \tag{2}$$

with A being the amplitude of the oscillating fluid, T the period of the oscillation and D the diameter of the cylinder. N_{KC} is a measure of the relative importance of the viscous drag forces compared to the close to potential flow inertia loads. For lower N_{KC} inertia dominates the force contribution. Keulegan and Carpenter (1958) carried out physical tank tests with regular waves passing a fully submerged, horizontally mounted cylinder.

Cases for horizontal cylinders are described by Dixon *et al.* (1979). Their objective was to calculate the vertical forces on a horizontally mounted cylinder in regular waves. They adapted the Morison equation and compared their computational results with measurements from physical tank tests. They found that in certain wave regimes the effects become more non-linear. For different combinations of wavelength and axis depth of the cylinder a doubling of the forces could be observed. This could not be described accurately by their mathematical approach.

In Section 3 results for horizontal cylinders with three levels of submergence are presented according to the experimental setup used by Dixon *et al.* (1979). In the first simulation the horizontal cylinder is placed with the axis at the free surface, being half-submerged. Next, the structure is positioned deeper in the water only showing 25% above still water level. Finally, the cylinder sits fully submerged but close to the water surface in the tank or computational domain.

Different non-linear effects are reported for vertical cylinders in waves. In the early 1990s model tests in Norway revealed that high nonlinear excitation of deepwater dynamically sensitive large volume structures was possible - this became known as "ringing". In certain wave conditions, especially when the wave height is approximately equal to the cylinder diameter, it breaks behind the cylinder when the crest passes the structure and thereby induces a secondary loading on the structure. This can influence the structure in such a way that it oscillates even after the wave has vanished. This so called “ringing” may damage the structure more that the impact of the wave itself. This highly non-linear effect is described by Chaplin *et al.* (1997) and Rainey (2007), who carried out tank tests to reproduce this phenomenon. They investigated a vertical cylinder in steep focused waves.

Section 4 deals with vertical cylinders in regular waves. Physical experiments described in Kriebel (1998) are reproduced numerically here, and the horizontal forces on the cylinder during the wave cycle are explored. Here the cylinder is large but still in the inertia regime. Further results are presented for a slender cylinder in steep regular waves to show the secondary load cycle.

1. Numerical Methods

1.1. Governing equations

Two commercial CFD packages are used in this work, Ansys CFX 11 (Ansys , 2006) and STAR-CCM+ by CD-Adapco (CD-Adapco , 2008). Both CFD solvers use the Navier-Stokes equations discretised on a 3-D mesh in order to calculate the velocities and pressures in the flow field. The flow is described by the equation of mass conservation as

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0, \tag{3}$$

and the three momentum conservation equations as

$$\frac{\partial u\rho}{\partial t} + \mathrm{div}\,(\rho u \boldsymbol{u}) = -\frac{\partial p}{\partial x} + \mathrm{div}\,(\mu \mathrm{grad}\ u) + S_x, \tag{4}$$

$$\frac{\partial v\rho}{\partial t} + \mathrm{div}\,(\rho v \boldsymbol{u}) = -\frac{\partial p}{\partial y} + \mathrm{div}\,(\mu \mathrm{grad}\ v) + S_y \tag{5}$$

and

$$\frac{\partial w\rho}{\partial t} + \mathrm{div}\,(\rho w \boldsymbol{u}) = -\frac{\partial p}{\partial z} + \mathrm{div}\,(\mu \mathrm{grad}\ w) + S_z. \tag{6}$$

where u, v and w are the components of the velocity vector $\boldsymbol{u}$ in the x, y and z directions, respectively. ρ is the fluid density, p the pressure, μ the fluid viscosity and t is time. S_i, with i being x, y or z, is the source term acting in i-direction, in which gravity forces are included. In integral form equations (3), (4), (5) and (6) can be rewritten as the general transport equation

$$\int_{\Delta t} \frac{\partial}{\partial t}\left(\int_{CV} \rho\phi d\mathcal{V}\right) dt + \int_{\Delta t}\int_{\mathcal{A}} \boldsymbol{n}.\,(\rho\phi\boldsymbol{u})\, d\mathcal{A}dt =$$

$$\int_{\Delta t}\int_{\mathcal{A}} \boldsymbol{n}.\,(\Gamma \mathrm{grad}\ \phi)\, d\mathcal{A}dt + \int_{\Delta t}\int_{CV} S_\phi d\mathcal{V}dt, \tag{7}$$

which is the starting point for discretising them either using the Control-Volume Finite Element method (CV-FE) or a Finite Volume method (FVM). Here, ϕ is the transported flow property such as velocity or pressure, Γ is the diffusion coefficient, $\mathcal{A}$ is the surface area of the control volumes (CV) face, $\mathcal{V}$ is the volume of the CV. S_ϕ is the source term and $\boldsymbol{n}$ is the outward-facing normal vector to a CV face.

Key differences between the FVM and CV-FE methods as implemented in the commercial CFD packages relate to the discretisation of the governing equations, the solution method and the free surface scheme. Both methods solve the Navier Stokes equations, (3) - (6), and use the Volume of Fluid (VoF) method for the free surface, solving the equations in both air and water and capturing the interface between them. In the CV-FE method, the equations are discretised using a control-volume based Finite Element approach (Ansys , 2006) that uses a shape function description of variables and a coupled solver, whereas the FVM applies surface and volume integrals to obtain cell centre values (CD-Adapco , 2008) and uses a segregated iterative solver.

1.2. Finite Volume Method (FVM)

The domain, here a numerical wave tank (NWT) and the structure such as a cylinder at the water surface, is subdivided into discrete volumes. The surface and volume integrals (7) performed on the control volumes are used to calculate the variable values at the centre node of the CV. This approach makes the Finite Volume method conservative by construction.

In discrete form for a cell-centred control volume with centre node 0 the general transport equation becomes

$$\frac{\partial}{\partial t}\left(\rho\phi\mathcal{V}\right)_0 + \sum_f \left(\rho\phi\boldsymbol{u}\boldsymbol{n}\right)_f = \sum_f \left(\Gamma \mathrm{grad}\ \boldsymbol{n}\right)_f + \left(S_\phi\mathcal{V}\right)_0 . \qquad (8)$$

The transient term in (8) is solved via a second-order scheme using the solution at the current time as well as those from the previous two. The advective term is solved by a second-order upwind scheme. Here the face values for the convective fluxes are interpolated linearly from the cell values of the surrounding cells.

To compute the gradients two methods are used for the FVM solver. The pressure gradients are computed using the weighted least squares method, which for a cell 0 can be expressed as

$$(\mathrm{grad}\phi)_r^u = \left[\sum_f \frac{(x_n - x_0) \otimes (x_n - x_0)}{(x_n - x_0) \cdot (x_n - x_0)}\right]^{-1} \left[\sum_f \frac{(\phi_0 - \phi_n)\,(x_n - x_0)}{(x_n - x_0) \cdot (x_n - x_0)}\right] \quad (9)$$

where x_0 and x_n represent the centroids of cell 0 and the neighbour cell respectively. f stands for a control volume face and ϕ_0 and ϕ_n represent the data values for cell 0 and its neighbour, here the pressure. For all other variables the gradients are approximated using Gauss' method as described by

$$(\mathrm{grad}\phi)_r^u = \frac{1}{\mathcal{V}_0} \sum_f \phi_f \boldsymbol{a}_f \quad (10)$$

where ϕ_f is the arithmetic average of the adjacent cells. To avoid non-physical behaviour the reconstruction gradients are limited. This might occur when the face values are calculated from the gradients and the reconstruction values exceed the cell values.

The FVM uses the well-known segregated, iterative scheme SIMPLE (Baliga and Patankar , 1980) to solve the Navier-Stokes equations. For capturing the free surface, the scheme developed by Ubbink (1997) and enhanced by Muzaferija and Perić (1999) is used, which requires a smaller time step than used by the coupled solver in the CV-FE method. The free surface reconstruction scheme does not smear the interface.

The free surface is calculated using a Volume of Fluid approach (VoF) in combination with a second-order high resolution interface capturing scheme. This results in an additional equation for each fluid phase, which needs to be solved. The change of the volume fraction c of one phase is governed by the transport equation

$$\frac{\partial}{\partial t}\int_{CV} c \, d\mathcal{V} + \int_{\mathcal{A}} c\,(\boldsymbol{u} - \boldsymbol{u}_f) \cdot \boldsymbol{a} \, d\mathcal{A}. \tag{11}$$

For a CV that is fully filled with one fluid c is 1, if no fraction of that fluid is present c is 0. This means that c is bound between 0 and 1 and the free surface lies in a cell that is partially filled with both fluids. The exact position of the free surface is reconstructed using the scheme proposed by Ubbink (1997) and Muzaferija and Perić (1999), which does not smear the interface as a first-order upwind scheme would and also does not produce over- or undershoots. It takes into account the orientation of the fluid interface and the local Courant number Co,

$$Co = \frac{\boldsymbol{u} \cdot \boldsymbol{n} \; S_f \Delta t}{\Delta \mathcal{V}_c}. \tag{12}$$

To avoid over- or undershoots the approximation of the cell-face value is limited using a normalized variable formulation, which ensures that the computed value lies within the grey shaded area in the Normalized Variable Diagram (NVD), as shown in Figure 1. Effectively this scheme blends between central differencing (CD), upwind differencing (UD), downwind differencing (DD) and lower upwind differencing (LUD) to ensure boundness of the variable and stability of the calculation at the same time.

To calculate the cell-face values c_f the following corrections are applied to ensure it is bound between 0 and 1:

$$\widetilde{c}_f = \begin{cases} \widetilde{c}_C & \text{if} \quad \widetilde{c}_C < 0 \\ 2\widetilde{c}_C & \text{if} \quad 0 \leq \widetilde{c}_C < 0.5 \\ 1 & \text{if} \quad 0.5 \leq \widetilde{c}_C < 1 \\ \widetilde{c}_C & \text{if} \quad 1 \leq \widetilde{c}_C \end{cases} \tag{13}$$

$$\widetilde{c}_f^* = \begin{cases} \widetilde{c}_f & \text{if} \quad Co < Co_l \\ \widetilde{c}_C + (c_f - c_C)\frac{Co_u - Co}{Co_u - Co_l} & \text{if} \quad Co_l \leq Co < Co_u \\ \widetilde{c}_C & \text{if} \quad Co_u \leq Co \end{cases} \tag{14}$$

$$\widetilde{c}_f^{**} = \widetilde{c}_f^* \sqrt{\cos\theta} + \widetilde{c}_C \left(1 - \sqrt{\cos\theta}\right) \tag{15}$$

By using the Courant number the scheme ensures that only such an amount of fluid leaves a cell as can be accommodated by the the acceptor cell or vice versa. θ is the angle between the normal of the fluid interface and the cell-surface vector $\boldsymbol{n}$. Finally the cell-face value is calculated as

$$c_f = \widetilde{c}_f^{**}\left(c_D - c_U\right) + c_U. \tag{16}$$

Subscripts U, C and D denote the upstream, cell-centre and downstream cell values. Co_l and Co_u are the lower and upper limits of the Courant number. For values $Co < Co_l$ HRIC is used, for $Co_l < Co < Co_u$ HRIC and UD are blended and for $Co_u < Co$ UD is used.

1.3. Control-Volume Finite Element Method (CV-FE)

The control-volume Finite Element approach combines the Finite Volume method considering the control volumes and the Finite Element method by using shape functions and finite element discretisation. The shape functions are used to calculate the change of a variable across the CV Ansys (2006); Baliga and Patankar (1983, 1980).

Figure 2 shows a 2D mesh section with unit depth. All nodes are surrounded by element surfaces that define a control volume around it, which is represented by the grey shaded area. All solution and fluid properties are stored in the mesh nodes. To solve the general transport equation (7) the terms have to be discretised on the mesh, i.e. the volumetric and surface flow terms have to be approximated over the control volume faces to obtain a set of linear equations, which can be solved. The volume terms are converted into their discrete form by approximating specific values in each sector and then integrating them over the control volume surface. The surface terms are obtained by first approximating the fluxes at the integration points (see Figure 2) from where the flows are calculated by integrating the fluxes over the appropriate control volume surface segment.

To overcome pressure-velocity decoupling on a non-staggered grid, a similar scheme as described by Rhie and Chow (1982) and modified by Majumdar (1988) is applied.

Although the solution variables are saved in the mesh nodes, it is necessary to calculate gradients at the integration points. Here the CV-FE solver combines the FE with the FVM method by making use of shape functions, which are typical for the FE approach. Across a mesh element a general flow variable ϕ varies as described by

$$\phi = \sum_{i=1}^{N_{Node}} N_i \phi_i, \tag{17}$$

with N being the shape function for node i and ϕ_i is the value of ϕ at node i. The shape functions are used to calculate geometric quantities, such as the coordinates of the integration points and surface area vectors $\vec{n}$. Also the summation of the shape function over all element nodes gives unity. Figure 3 shows the geometry of the shape function for a hexahedral control volume. N_i is given by

$$N_1\left(r,s,t\right) = \left(1-r\right)\left(1-s\right)\left(1-t\right), \tag{18}$$

$$N_2\left(r,s,t\right) = r\left(1-s\right)\left(1-t\right),$$

$$N_3\left(r,s,t\right) = rs\left(1-t\right),$$

$$N_4\left(r,s,t\right) = \left(1-r\right)s\left(1-t\right),$$

$$N_5\left(r,s,t\right) = \left(1-r\right)\left(1-s\right)t,$$

$$N_6\,(r,s,t) = r\,(1-s)\,t,$$

$$N_7\,(r,s,t) = rst$$

and

$$N_8\,(r,s,t) = (1-r)\,st.$$

As in a standard FE approach, the shape functions are used to evaluate the spatial derivatives of the diffusion terms and the pressure gradients. The discretisation of the advection terms uses a high resolution scheme similar to the one described by Barth and Jesperson (1989).

The CV-FE method uses the Barth and Jesperson (1989) scheme to reconstruct the free surface from the VoF results. This scheme is dependent on the filling level of the surrounding cells rather than the Courant number as in the Ubbink (1997) method used by the FVM, and so not timestep dependent. Additionally the solver used by the CV-FE method is fully coupled, meaning that all equations are solved in one large matrix at once (Zwart , 2005; Zwart *et al.*, 2003). To take advantage of these properties, the timestep may be chosen to be relatively large compared to a segregated solver.

2. Wave-wave interaction in focused wave groups

In this work, we follow the wave tank geometry and set up used in physical experiments described in Ning *et al.* (2009). They used a wave tank with plan dimensions of 69 m x 3 m and a water depth of 0.5 m. The waves were generated by a piston wavemaker and wave reflections were absorbed by a 4 m foam layer placed at the downstream end of the flume. Wave gauges (WG) were used to measure the surface elevation around the point of the maximum wave elevation and the layout is shown in Figure 4. In the study by Ning *et al.* (2009), four NewWave cases are investigated with different input amplitudes; here we reproduce numerically cases 2, 3 and 4.

The NewWave theory describes the surface elevation and wave velocity components of a focused group of localised waves derived from a measured or theoretical spectrum, such as JONSWAP or Pierson-Moskowitz. The waves are superposed and brought into phase at one point in the tank at a specified time. This generates an extreme wave event, which represents the wave environment of the underlying spectrum. By increasing the applied maximum focused wave height, wave breaking at a defined location can be achieved.

In the physical experiment the focus point was set to 11.4 m downstream of the wavemaker, which is the position of wave gauge no. 5. The distances to the other wave gauges, measured from WG 5, may be taken from Table 1.

2.1. Domain

For both software packages similar domains are used, as shown in Figure 4. To save computational resources the domain is shortened to 13 m, following the approach taken by Ning *et al.* (2009) in their fully non-linear potential flow simulations. The entire domain is 1m high with a water depth of 0.5 m. Between $x = 10$ m and $x = 13$ m, a damping layer is installed which prevents reflections from the right-hand boundary. In this area the dynamic viscosity of the water fraction increases linearly from 8.94 x 10^{-4} Pa s up to 1600 Pa s.

The left-hand boundary is a velocity inlet, where the horizontal and vertical velocity components are applied together with the volume fraction of air and water. These are calculated following the derivations described by Dalzell (1999), which are extended to the number of wave components N required for this experiment (see section 2.2). The velocities are applied for the water fraction only and the velocity of the air fraction at the inlet boundary is set to zero. The top boundary is a pressure outlet, allowing only air to leave or enter the domain. The remaining boundaries at the bottom and right-hand side are walls. No turbulence model is applied.

Although a piston wavemaker is used in the physical experiment, wave height data is extracted at wave gauge 5, located 11.4 m downstream from the wavemaker. It is assumed that when the wave has travelled to gauge 5, it will have developed and be reasonable approximated by Stokes theory. Applying piston wavemaker characteristics would only be accurate, if the entire length of the wave tank were modelled, but here only a shortened part of the tank has been modelled in order to conserve the computational cost.

The vertical coordinates where the water volume fraction is equal to 0.5 are taken to be the position of the free surface and are extracted at the positions of the wave gauges for every time step. These are given as surface elevation time history plots shown in section 2.3.

The CV-FE calculations are carried out on a uniform hexahedral mesh. Simulations are done on a pseudo 2D mesh with one cell layer thickness. For all NewWave cases the same mesh is used. Grid convergence studies have been reported in previous work by the authors, Westphalen *et al.* (2007) and Westphalen *et al.* (2008), and found that the number of cells needed in the vertical direction is 110 over the entire domain height. Thus for case 2 the wave height is resolved by 14 cells in the vertical direction, this is sufficient to resolve the free surface accurately. For case 3 and 4 with $A = 0.0875$ m and 0.1031 m this gives 9.6 and 11.3 cells to resolve the wave height respectively.

The CV-FE approach uses the Barth and Jesperson (1989) scheme to reconstruct the free surface from the VoF results. This scheme is not timestep dependent. Additionally the solver is fully coupled, meaning that all equations are solved in one large matrix at once (Zwart , 2005; Zwart *et al.*, 2003). To take advantage of these properties, the timestep may be chosen to be relatively large compared to a segregated solver. For cases 2 and 3 the timestep is set to 0.01 s and for case 4 is 0.005 s. For all CV-FE cases, high performance computing is used and the simulations are run in parallel mode using 16 processors.

The FVM method uses the well-known segregated, iterative scheme SIMPLE to solve the Navier-Stokes equations. For capturing the free surface the scheme developed by Ubbink (1997) and enhanced by Muzaferija and Perić (1999) is used, which requires a smaller time step than used by the CV-FE method to ensure the Courant number is less than or equal to 1.0. The calculation for case 2 is made with a timestep of 0.001 s. Case 3 and case 4 are started with the same timestep for the first 5 s of the simulation and then the timestep is reduced to 0.00025 s. FVM simulations are run in parallel mode on a modern desktop PC with Intel Pentium Duo Core Processors, each 2.4 GHz, and 2 Gb RAM.

The calculations are carried out on a hexahedral mesh, that is refined around the free surface. Due to the increase in the expected wave height, the refined area is vertically extended from case 2 to case 4. Additionally the cells at the inlet are refined for better definition of the velocity field and the rapidly moving free surface during the calculation. The refined area ends at $x = 4.5$ m. Beyond that point the mesh is uniform having the same cell size as the regions above and below the refined region. The calculation for case 2 is made with a timestep of 0.001s. Case 3 and case 4 are started with the same timestep as case 2 for the first 5s of the simulation. After that the timestep is reduced to 0.00025 s. These settings ensure a Courant number smaller than or close to 1.

2.2. Generation of focused wave groups (NewWave)

The concept of the NewWave formulation is to generate several waves of relatively small amplitudes and different periods. These waves interact and constructively interfere to build up a localised extreme wave, larger than any individual wave created at the paddle, focused at a specified position and time in the tank. In the numerical calculations the waves are generated from the spectra shown in Figure 5. These spectra are obtained by Fourier transformation of the free surface data measured in the experiments as described by Ning *et al.* (2009) and Zang *et al.* (2006).

For each wave component n the amplitude a_n is defined as

$$a_n = A \frac{S_n\left(f\right) \Delta f}{\sum_n S_n\left(f\right) \Delta f} \tag{19}$$

where $S(f)$ is the spectral density, Δf is the frequency step depending on the number of wave components N and bandwidth. A is the target linear amplitude of the focused wave. Thus, the amplitude of every spectral component in the NewWave group sclaes as the power density within that frequency band in the assumed sea-state. Equivalently, NewWave is simply the scaled auto-correlation function corresponding to a specified frequency spectrum such as JONSWAP, Pierson-Moskowitz etc. The properties for each case considered can be seen in Table 2.

The underlying equations, from which the signal of the physical wave-maker is derived, come from second order Stokes theory and are given by Ning *et al.* (2009) and Dalzell (1999). The input signal for the CFD runs is the sum of the first $^{(1)}$ and the second $^{(2)}$ order component for the horizontal and vertical water velocity component u and w and the surface elevation η:

$$\eta = \eta^{(1)} + \eta^{(2)}, \tag{20}$$

$$u = u^{(1)} + u^{(2)} \tag{21}$$

and

$$w = w^{(1)} + w^{(2)}. \tag{22}$$

The first order component for the surface elevation is

$$\eta^{(1)} = \sum_{i=1}^{N} a_i \cos \psi_i \tag{23}$$

with ψ_i as

$$\psi_i = k_i x - \omega_i t + \epsilon_i. \tag{24}$$

The first order velocities $u^{(1)}$ and $w^{(1)}$ are

$$u^{(1)} = \sum_{i=1}^{N} \frac{a_i k_i g}{\omega_i} \frac{\cosh k_i (z+h)}{\cosh k_i h} \cos \left(k_i (x - x_0) - \omega_i (t - t_0) + \epsilon_i\right) \tag{25}$$

and

$$w^{(1)} = \sum_{i=1}^{N} \frac{a_i k_i g}{\omega_i} \frac{\sinh k_i (z+h)}{\cosh k_i h} \sin \left(k_i (x - x_0) - \omega_i (t - t_0) + \epsilon_i\right). \tag{26}$$

The second order parts for the surface elevation $\eta^{(2)}$ and the velocities $u^{(2)}$ and $w^{(2)}$ are given by

$$\eta^{(2)} = \sum_{i=1}^{N} \sum_{j=i+1} a_i a_j B_{i,j}^{+} \cos(\psi_i + \psi_j) + \sum_{i=1}^{N} \sum_{j=i+1} a_i a_j B_{i,j}^{-} \cos(\psi_i - \psi_j)$$

$$+\sum_{i=1}^{N}\frac{a_j^2k_i}{4\tanh\left(k_ih\right)}\left[2+\frac{3}{\sinh^2\left(k_ih\right)}\right]\cos\left(2\psi_i\right)-\sum_{i=1}^{N}\frac{a_j^2k_i}{2\sinh\left(2k_ih\right)} \tag{27}$$

and

$$\phi^{(2)}=\sum_{i=1}^{N}\sum_{j=i+1}a_ia_jA_{i,j}^{+}\frac{\cosh\left(\left(k_i+k_j\right)\left(z+h\right)\right)}{\cosh\left(\left(k_i+k_j\right)h\right)}\sin\left(\psi_i+\psi_j\right)$$

$$+\sum_{i=1}^{N}\sum_{j=i+1}a_ia_jA_{i,j}^{-}\frac{\cosh\left(\left(k_i-k_j\right)\left(z+h\right)\right)}{\cosh\left(\left(k_i-k_j\right)h\right)}\sin\left(\psi_i-\psi_j\right)$$

$$+\sum_{i=1}^{N}a_i^2\frac{3\omega_i}{8}\frac{\cosh\left(2k_i+k_jh\right)}{\sinh^4\left(k_ih\right)}\sin\left(2\psi_i\right), \tag{28}$$

with components of the interaction kernels $A_{i,j}^{+/-}$, $B_{i,j}^{+/-}$ and $D_{i,j}^{+/-}$ defined as

$$A_{i,j}^{+}=-\frac{\omega_i\omega_j\left(\omega_i+\omega_j\right)}{D_{i,j}^{+}}\left[1-\frac{1}{\tanh\left(k_ih\right)\tanh\left(k_jh\right)}\right]$$

$$+\frac{1}{2D_{i,j}^{+}}\left[\frac{\omega_i^3}{\sinh^2\left(k_ih\right)}+\frac{\omega_j^3}{\sinh^2\left(k_jh\right)}\right] \tag{29}$$

$$A_{i,j}^{-}=\frac{\omega_i\omega_j\left(\omega_i-\omega_j\right)}{D_{i,j}^{-}}\left[1+\frac{1}{\tanh\left(k_ih\right)\tanh\left(k_jh\right)}\right]$$

$$+\frac{1}{2D_{i,j}^{-}}\left[\frac{\omega_i^3}{\sinh^2\left(k_ih\right)}-\frac{\omega_j^3}{\sinh^2\left(k_jh\right)}\right] \tag{30}$$

$$B_{i,j}^{+}=\frac{\left(\omega_i^2+\omega_j^2\right)}{2g}-\frac{\omega_i\omega_j}{2g}\left[1+\frac{1}{\tanh\left(k_ih\right)\tanh\left(k_jh\right)}\right]$$

$$\cdot\left[\frac{\left(\omega_i+\omega_j\right)^2+g\left(k_i+k_j\right)\tanh\left(\left(k_i+k_j\right)h\right)}{D_{i,j}^{+}}\right]$$

$$+\frac{(\omega_i+\omega_j)}{2gD^+_{i,j}}\left[\frac{\omega_i^3}{\sinh^2(k_ih)}+\frac{\omega_j^3}{\sinh^2(k_jh)}\right] \tag{31}$$

$$B^-_{i,j}=\frac{\left(\omega_i^2+\omega_j^2\right)}{2g}+\frac{\omega_i\omega_j}{2g}\left[1-\frac{1}{\tanh(k_ih)\tanh(k_jh)}\right]$$
$$\cdot\left[\frac{(\omega_i-\omega_j)^2+g(k_i-k_j)\tanh((k_i-k_j)h)}{D^-_{i,j}}\right]$$
$$+\frac{(\omega_i-\omega_j)}{2gD^-_{i,j}}\left[\frac{\omega_i^3}{\sinh^2(k_ih)}-\frac{\omega_j^3}{\sinh^2(k_jh)}\right] \tag{32}$$

$$D^+_{i,j}=(\omega_i+\omega_j)^2-g(k_i+k_j)\tanh((k_i+k_j)h) \tag{33}$$

and

$$D^-_{i,j}=(\omega_i-\omega_j)^2-g(k_i-k_j)\tanh((k_i-k_j)h) \tag{34}$$

The second order velocity components are obtained by the relevant spatial derivatives of the second order potential $\phi^{(2)}$ to be differentiated with respect to x and z

The wavemaker is located at $x = 0$ m; x_0 is the focus point, which is set to 3 m for case 2 and to 3.27 m for cases 3 and 4; t is the time; t_0 is the focus time, which is 9.2 s for case 2 and 10s for cases 3 case 4. Also in the formulae are the wavenumber k_i, the frequency ω_i , the phase angle ϵ_i (which is set to 0 for the calculations), the water depth h, vertical position z and the number of wave components N.

In this work, the incoming wave entering the computational domain is fluxed in through a transparent boundary. This flux is defined in terms of either linear theory, or linear theory with second order corrections.

2.3. NewWave generation results

For this paper crest and trough focused wave groups with three different levels of linearity or non-linearity were simulated. 16 simulations using the FVM and CV-FE solvers were performed to compare the surface elevations at the focus point with the experimental results by Ning *et al.* (2009). For each solver results are shown corresponding to input waves defined using linear and linear plus second order theory. The surface elevations are non-dimensionalised in terms of the wave crest of the target NewWave A, and time is defined in terms of the appropriate wave period, as shown in Table 2. Table 3 shows the maximum surface elevations for these cases. As the actual focus point and time differ for every simulation, which has also been reported by Baldock et al. (1996), all graphs are shifted in time to coincide with one another. The graphs are adjusted at $t/T = 0$, which is the time when the maximum surface elevation occurs. Figures 6 - 8 show the water surface elevation from the two solvers compared with the measured results from the physical experiment for all crest focused cases 2, 3 and 4.

For case 2, the case of the weakest non-linearity, the FVM with first order wave input signal does not reach the required height. The simulation including the second order wave components, however, overestimates the crest elevations by approximately 24%, with a value of 0.015 m for an input wave crest of 0.0632 m. The trough elevations for the surrounding troughs improve from first order to first plus second order wave setup for the FVM solver in terms of matching with the physical experiments. The CV-FE solver predicts surface elevations for both wave signals slightly higher than those measured in the physical tank tests. Here the troughs that surround the central wave are higher and do not exactly coincide with the results of the experimental results, although one can see an improvement from first order to second order wave signal.

The numerical results for case 3 generally show the best agreement with the physical experiments; particularly the crest elevations are predicted with only a slight difference for both input signals. Moving from first to second order wave input also improves the trough elevations.

Case 4 is the test with the steepest wave, which almost broke in the physical experiment. The numerical results are very good for the maximum crest elevations. However, the surrounding wave train does not agree as well as the previous tests, though the trends are the same. The wave preceding the central wave is much larger than the measured values for both codes, but still smaller and not symmetric to the wave that follows the main wave, as it can also be observed in Figures 6 and 7. The strong asymmetry around the main wave crest is predicted less well for steeper wave cases, and this is evident for both FVM and CV-FEM methods, however, the peak is predicted well for all cases.

The actual focus point in the numerical calculations lies further downstream than specified, as can be seen from Table 3. Also, it is further away for the simulations with higher order wave signal.

A useful tool to assess the non-linearity of results is the comparison of the sum and difference plots. Therefore additional simulations were carried out, with trough focused instead of crest focused waves. By subtracting and summing the crest and trough focused wave time histories (and dividing them by two), as described by Zang *et al.* (2006), the plots for cases 2 and 4, shown in Figure 9, are obtained. By subtracting the signals from one another the the linear part of the solution plus the odd harmonics is obtained (solid line) and by adding the signals, the even harmonics of the wave time history are extracted (dashed line). As it is expected the non-linearities increase from case 2 to case 4, which has a greater NewWave target amplitude. The dashed line in Fig. 9b for case 4 has a greater amplitude and is less smooth than that in Fig. 9a for case 2. Similarly, the odd harmonic component for case 4 has a spiked peak typical for higher order effects.

3. Fixed horizontal cylinder in regular waves

3.1. Computational domain

For the numerical simulations 3-dimensional meshes containing mostly hexahedral cells are used. However, as the domain is very thin the simulation is in fact 2-dimensional. It is 10 m long, 2 m high and has a width of 0.1 m. The cylinder axis is perpendicular to the plane of the domain and sits one wavelength λ downstream of the inlet and is defined as a wall. The diameter of the cylinder D is 0.25 m. In both codes the bottom and far side boundary are walls as well. The sides are set up as symmetry boundaries. The waves are generated at the velocity inlet on the left-hand side. The top boundary is a pressure outlet with only air being allowed to leave or enter the domain. The general arrangement is similar to those of the NewWave simulations described previously (see Fig. 4).

Meshes are generated using the automatic mesh generation packages and Cartesian hexahedral cells are used as the basis for wave and wave-structure interaction simulations as these work best with VOF schemes. Where a cylinder is included in the domain, hexahedral cells are deformed around the cylinder in the CV-FE method, whereas in the FVM method, the cylinder is cut out of the background mesh and has a prism layer of cells around it to reduce the influence of deformed cells and optimise the computation of boundary flows. Portions of the meshes close to the cylinder are shown in Figures 10 and 11. For the three horizontal cylinder cases presented here, the CV-FE meshes contain 79495, 69537 and 69537 cells and a timestep of 0.005 s was used; the FVM meshes have 113856, 113606 and 114599 cells and are calculated with a timestep of 0.001 s.

The deformed cells around the cylinder occurring in the CV-FE method influence the initial water surface for some cases, notably for the numerical experiments where the cylinder is 3/4 and fully submerged. Nevertheless, as it takes some time for the water to travel down the numerical tank, this effect can be neglected because the water has enough time to settle and adjust itself to a physical behaviour before the wave arrives at the cylinder.

3.2. Generation of regular waves

The water depth h for all numerical simulations is 1 m, the wave period T is 1.646 s with wave amplitudes A being 0.125 m, 0.05 m and 0.075 m. The properties including the fixed displacement of the cylinder below still water level d, kA, kh and the Keulegan-Carpenter numbers N_{KC} for each case are shown in Table 4. According to the physical experiments by Dixon *et al.* (1979) the wave signal is accurate to first order. The waves are generated using the vertical and horizontal water velocity components u and w, with

$$u = \frac{gAk\ \cosh\left(k\left(z+h\right)\right)\cos\left(kx-\omega t\right)}{\omega\ \cosh\left(kh\right)} \tag{35}$$

and

$$w = -\frac{gAk\ \sinh\left(k\left(z+h\right)\right)\sin\left(kx-\omega t\right)}{\omega\ \cosh\left(kh\right)}. \tag{36}$$

These are applied for the water fraction only, which is defined using the surface elevation η from first order wave theory, with

$$\eta = A\cos\left(kx-\omega t\right). \tag{37}$$

A is the wave amplitude, g is the acceleration due to gravity, k is the wavenumber, h the water depth, z the vertical position and ω is the wave frequency. To avoid reflections from the far end boundary a damping zone is defined. This is done by increasing the dynamic viscosity, μ, of the water linearly from 8.8871 x 10^{-4} Pa s to 800.0 Pa s over the last 3 m of the domain.

3.3. Horizontal cylinder results

To compare the numerical results with those obtained by Dixon *et al.* (1979) the vertical forces F_z on the cylinder resulting from drag and pressure on the surface are extracted. The forces F' shown in all figures are non-dimensionalised using the following expression

$$F' = \frac{F_z}{g\rho\left(1/4\,\pi D^2 l\right)} \tag{38}$$

where F_z is the measured vertical force on the cylinder, ρ is the density of water, D is the cylinder diameter and l is the length of the cylinder. Figure 12 shows the non-dimensionalised vertical force time histories as predicted using each code compared to that measured experimentally for d = 0.0. For longer time simulations, there is likely to be reflection from the cylinder that would in turn reflect from the wavemaker and alter the generated wave. For this reason, results were extracted from simulation of only the first two steady-state waves interacting with the cylinder.

For the case $d = 0.0$ m the numerical predictions fit well with the measured ones. At the beginning of the wave cycle heave forces are dominant. Once past the instant of peak force, the force time history exhibits a saddle point and then becomes negative as the wave trough passes. When the wave passes further and the water level rises, the downward force reduces. This can also be seen in Figure 13, which shows the surface profiles around the cylinder at $d = 0.0$ m throughout the wave cycle, predicted using the FVM.

Figure 12 also shows the vertical forces for the horizontal cylinder, which is positioned with its axis 0.075 m below the still water level. Here the qualitative characteristics are the same as for the half-submerged cylinder but the actual values are much smaller.

The match between numerical simulation and physical experiment is very good.

The last set of numerical results, however, do not agree so well with the physical experiments. For the cylinder positioned at $d =$ -0.15 m, which makes the structure fully submerged, the differences for both numerical methods are significant. The forces have the correct phase relative to the wave motion and the numerical time histories are in reasonable agreement with the measurements presented by Dixon *et al.* (1979). In Figure 14, the surface profiles close to the cylinder predicted for the case d = -0.15 m throughout the wave cycle are shown. It is evident that the cylinder surface becomes partially exposed during the wave cycle.

4. Vertical cylinders in regular waves

This section describes the numerical simulations of a vertical cylinder in two regular wave environments. The first set of tests aims to reproduce the measured results of physical experiments by Kriebel (1998), who also compares the horizontal forces on a vertical circular cylinder in regular waves with computations using first and second order diffraction theory. From this publication two cases are chosen as described in Table 5.

4.1. Computational Domain

The numerical wavetanks for both solvers are identical. The simulations are performed in a 3-dimensional domain with the dimensions x, y and z equal to 12, 1.65 and 0.9 m. The still water level is 0.45 m. The diameter of the cylinder is 0.325 m as in the physical experiments. The centre of it is located 3.77 m downstream from the inlet, which is equal to 1 wavelength for the first wave setup and approximately 2 wavelengths for the second.

As with the horizontal cylinder cases (see Section 3) the waves are generated using the horizontal and vertical velocity components, which are applied underneath the appropriate surface elevations at the left hand side velocity inlet (see Section 3.2). The sides are symmetry planes and the top is a pressure outlet, with air being allowed to leave or enter the domain. The remaining boundaries are modelled as walls, i.e. the cylinder, the bottom and the far end boundary.

The meshes, however, are different for both solvers. The grid that is used for the CV-FE solver contains 570000 hexahedral cells. It is refined around the cylinder and the area where the free water surface is expected to travel upstream of the cylinder. Downstream of the cylinder the mesh is relatively coarse to save computational resources. The cell size around the free surface upstream of the cylinder is 0.011 m. The cylinder is discretised by 3240 quadrilateral elements. The number of cells covering the perimeter is 72, which is constant over the entire cylinder height. The refinement of the cylinder in the area where the waves hit the structure is in the vertical direction only. The timestep for the CV-FE solver is 0.005 s.

The mesh used for the FVM solver contains 870000 mostly hexahedral cells. The cells around the water surface area are isotropically refined to an edge length of 1.25 cm, which gives the necessary resolution of approximately 10 cells per wave height (Westphalen *et al.*, 2008). The cylinder itself is modelled with 3005 faces of which 37 are triangular, 2898 quadrilateral and 70 polygonal. The different cell types result from the meshing algorithm, which cuts the geometry out of the initial hexahedral mesh rather than using a body fitted grid as it is done for the CV-FE solver. At the top and the bottom the cylinder contains 52 faces around the perimeter. In the central region of the cylinder the mesh is refined not only in the vertical direction, as is done for the CV-FE solver, but also tangentially. Between the vertical positions of 0.35 m and 0.55 m,104 cells wrap the cylinder perimeter.

4.2. Results

The validation of the codes is done by comparing the total horizontal forces on the cylinder due to the waves with the measured results from Kriebel (1998). As described by Kriebel the forces are normalised with the analytic results from linear diffraction theory, which is given by

$$\frac{F}{F_0} = F \frac{kh}{\rho g a H h \tanh kh}. \tag{39}$$

F is the measured or extracted horizontal force on the cylinder, a the radius of the cylinder and H the wave height.

Figure 15 shows the comparison between the measured data and the numerical simulations for both codes and the two wave conditions described in Table 5. Both numerical methods show good agreement for both cases, slightly better for the second one. Small differences in the results are expected because the physical data is averaged over 10 wave periods, whereas the numerical results represent one wave cycle only. For the second case where N_{KC} is larger, i.e. the drag forces dominate, the physical experiments and the calculations reach a maximum value almost twice that predicted by linear diffraction theory.

4.3. Secondary load cycle

The aptly named "ringing" of vertical surface piercing cylinders in steep waves is a highly nonlinear effect. It was discovered during design work on deepwater concrete platforms in Norway in the early 1990s and is described by Stansberg (1997) and Grue and Huseby (2002). The ringing itself only occurs when the cylinder is mounted elastically, which is not the case for the simulations presented here. However, Chaplin *et al.* (1997) have carried out tank tests to reproduce this effect. First they studied fixed vertical cylinders in steep focused waves and then cylinders mounted elastically in order to measure the response. Thus they were able to identify secondary loading within the measured force curve. Chaplin *et al.* (1997) present the definition of the secondary load cycle reproduced in Figure 16a, where the horizontal force on the cylinder is plotted against time. They state that this effect occurs when the wave height is equal to the cylinder diameter. Using the FVM solver the horizontal force predicted for a slender cylinder of diameter equal to 0.1625 m in the same wave climate as described above can be seen in Figure 16b. The secondary load cycle is present for this wave, although it cannot be compared exactly with 14a as the input wave conditions are different.

5. Conclusions

This work describes an investigation of the suitability of the use of the numerical techniques implemented in commercial CFD packages for design of offshore structures under extreme waves. Simulations of extreme focused wave events and wave-structure interaction of regular waves with fixed horizontal and vertical cylinders are presented and the predicted results compared with physical experiment data. Comparison is drawn between predicted and measured wave surface elevation for wave-only cases. It is found, that use of second order theory to supply leading order approximations to the bound wave structure at the inlet to the computational domains leads to significantly improved agreement in comparison with linear wave theory. However, some asymmetry exists in the numerical predictions of surface elevation in the most extreme wave focussing case.

Simulations of fixed horizontal and vertical cylinders in regular waves are investigated to explore the loading on the structures. The vertical and horizontal forces are compared with physical experiments described by Dixon *et al.* (1979), Kriebel (1998) and Chaplin *et al.* (1997) and show very good agreement. In particular, the highly non-linear effects of the wave-structure interaction cases were picked up very satisfactorily. These are the double frequency force oscillation on the horizontal cylinder and the secondary load cycle for the vertical cylinder case, which may cause severe damage due to the ringing of the structure after being passed by the wave. Some differences remain in the prediction of loading on the initially submerged horizontal cylinder and inspection of the surface profiles for this case reveal that partial wetting and drying of the cylinder surface occurs during the wave cycle. Nevertheless, when run in parallel, the numerical techniques considered here and implemented in commercial CFD packages are demonstrated to be powerful tools for offshore structure design, and able to predict highly non-linear wave interaction.

Acknowledgements

The authors would like to thank Prof. P.H. Taylor from the University of Oxford for his contributions to the research and during the process of writing this paper. The work was funded by the Engineering and Physical

Science Research Council under the project title “Extreme Wave Loading on Offshore Wave Energy Devices using CFD: a Hierarchical Team Approach” (Grant No. EP/D077508).

References

Ansys Inc. (2006). ANSYS CFX-Solver Theory Guide. Canonsburg, USA.

Baldock, T. E. , Swan, C., Taylor, P. H.(1996). ”A laboratory study of nonlinear surface waves on water.” Phil. Trans. R. Soc. London A 354: 649-676.

Baliga, B. R., Patankar, S. V. (1980). ”A New Finite-Element Formulation for Convection-Diffusion Problems.” Numerical Heat Transfer, Part A 3: 393-409.

Baliga, B. R., Patankar, S. V. (1983). ”A Control Volume Finite-Element Method for Two-Dimensional Fluid Flow and Heat Transfer.” Numerical Heat Transfer, Part A 6: 245-261.

Barth, T. J., Jesperson, D. C.; (1989). The Design and Application of Upwind Schemes on Unstructured Meshes. AIAA. Reno/Nevada.

CD-Adapco (2008). STAR CCM+ Version 4.02.011 . London, UK.

Chaplin, J. R., Rainey, R. C. T., Yemm, R. W. (1997). ”Ringing of a vertical cylinder in waves.” Journal of Fluid Mechanics 350: 119-147.

Dalzell, J. F. (1999). ”A note on finite depth second-order wavewave interactions.” Applied Ocean Research 21: 105-111.

Dixon, A. G., Greated, C. A., Salter, S. H. (1979). "Wave Forces on Partially Submerged Cylinders." Journal of the Waterway, Port, Coastal and Ocean Devision 105(4): 421-438.

Easson, W. J., Greated, C. A., Duranni, T. S. (1985). "Force Spectra from Partially Submerged Circular Cylinders in Random Seas." Journal of Waterway, Port, Coastal and Ocean Engineering 111(5): 856-879.

Ferziger, J. H., Perić, M.(2001). Computational Methods for Fluid Dynamics Heidelberg, Springer.

Grue, J., Huseby, M. (2002). "Higher-harmonic wave forces and ringing of vertical cylinders." Applied Ocean Research 24: 203-214.

Hogben, N., Miller, B. L. , Searle, J. W., Ward, G. (1977). "Estimation of fluid loading on offshore structures." Institute of Civil Engineering 63(Part 2): 515-562.

Keulegan, G. H., Carpenter, L. H. (1958). "Forces on Cylinders and Plates in an Oscillating Fluid." Journal of Research of the National Bureau of Standards 60(5): 423-440.

Kriebel, D. L. (1998). "Non-linear wave interaction with a vertical circular cylinder: wave forces" Ocean Engineering 25(7): 597-605.

Longuet-Higgins, M. S. (1977). The Mean forces Exerted by Waves on Floating or submerged Bodies with Application to Sand Bars and Wave Power

Machines. Proceedings of the Royal Society of London. Series A, Mathematical and Physical Sciences.

Ma, Q.W. and Yan, S 2009. "QALE-FEM for Numerical Modelling of Nonlinear Interaction between 3D Moored Floating Bodies and Steep Waves," International Journal for Numerical Methods in Engineering, Vol. 78, pp. 713-756.

Majumdar, S. (1988). "Role of underrelaxation in momentum interpolation for calculation of flow with nonstaggered grids." Numerical Heat Transfer, Part A 13: 125-132.

Morison, J. R., O'Brien, M. P., Johnson, J. W., Shaaf, S. A.(1950). The Force Exerted by surface Waves on Piles. Petroleum Transactions, American Institute of Mining and Petroleum Engineers. 189: 149-154.

Muzaferija, S. Perić, M. (1999). Computation of free-surface flows using interface-tracking and interface-capturing methods. Nonlinear Water Wave Interaction. O. M. Mahrenholtz, M. Hamburg, WIT Press. 24: 252.

Newman, J. N., Lee, C.-H. (2002). "Boundary-element methods in offshore structure analysis." Journal of Offshore Mechanics and Arctic Engineering 124: 81-89.

Ning, D. Z., Zang, J., Liu, S. X., Eatock Taylor, R. , Teng, B. , Taylor, P. H. (2009). "Free Surface and Wave Kinematics for Nonlinear Focused Wave Groups." Ocean Engineering 36(15-16): 1226-1243.

Ogilvie, T. F. (1963). "First and Second Order Forces on a Cylinder Submerged Under a Free Surface." Journal of Fluid Mechanics 16: 451-472.

Rainey, R. C. T. (2007). "Weak or strong nonlinearity: the vital issue." Journal of Engineering Mathematics, Newman Honorary Volume.

Rhie, C. M., Chow, W. L. (1982). A numerical study of the turbulent flow past an isolated airfoil with trailing edge separation. AIAA/ASME 3rd Joint Thermophysics, Fluids, Plasma and Heat Transfer Conference, St. Louis, Missouri.

Sarpkaya, T., Issacson, M. (1981). Mechanics of Wave Forces on Offshore Structures, Van Nostrand Reinhold.

Schäffer, H. A. (1996). "Second-order wavemaker theory for irregular waves." Ocean Engineering 23(1): 47-88.

Stansberg, C. T. (1997). Comparing ringing loads from experiments with cylinders of different diameters - An empirical study. 8th Conference on the Behaviour of Offshore Structures (BOSS'97) Pergamon/Elsevier.

Ubbink, O. (1997). Numerical prediction of two fluid systems with sharp interfaces. Department of Mechanical Engineering. London, Imperial College of Science, Technology & Medicine. PhD: 138.

Westphalen, J., Greaves, D. M., Williams, C. J. K. (2007), Comparison of Free Surface Wave Simulations using STAR CCM+ and CFX, Numerical Towing Tank Symposium, Hamburg 23.-25. Sep. '07, Westphalen, J., Greaves, D. M., Williams, C. J. K, Zang, J., Taylor, P. H. (2008). Numerical Simulation of Extreme Free Surface Waves. 18th International Offshore and Polar Engineering Conference, Vancouver, Canada.

Yan, S. and Ma, Q.W. (2008). "Nonlinear simulation of 3d freak waves using a fast numerical method". In JS Chung, ST Grilli, S Naito, and Q Ma, editors, Proceedings of the 18th International Offshore and Polar Engineering Conference, Vol 3.

Yan, S. and Ma, Q.W. (2010). "QALE-FEM for modelling 3D overturning waves," International Journal for Numerical Methods in Fluids, Vol. 63, pp.743 - 768

Zang, J., Gibson, R.; Taylor, P.H.; Eatock Taylor, R.; Swan, C. (2006). "Second Order Wave Diffraction Around a Fixed Ship - Shaped Body in Unidirectional Steep Waves." Journal of Offshore Mechanics and Arctic Engineering 128: 10.

Zwart, P. J. (2005). Numerical Modelling of Free Surface and Cavitating Flows. VKI Lucture Series, Ansys Canada Ltd.: 25.

Zwart, P.J., Scheuerer, M., Bogner, M.(2003). Free Surface Modelling of an Impinging Jet. ASTAR International Workshop on Advanced Numerical Methods for Multidimensional Simulation of Two-Phase Flow. Garching, Germany.