Proceedings of the 30th International Conference on Ocean, Offshore and Arctic Engineering
OMAE2011
June 19-24, 2011, Rotterdam, The Netherlands

OMAE2011-49488

EXTREME WAVE IMPACT ON OFFSHORE PLATFORMS AND COASTAL CONSTRUCTIONS

Arthur E.P. Veldman*
Roel Luppes
Institute for Mathematics and Computer Science
University of Groningen
P.O. Box 407, 9700 AK Groningen
The Netherlands
{a.e.p.veldman, r.luppes}@rug.nl

Tim Bunnik
MARIN
P.O. Box 28, 6700 AA Wageningen
The Netherlands
t.bunnik@marin.nl

René H.M. Huijsmans
Bulent Duz
Department of Ship Hydrodynamics
Technical University of Delft
Mekelweg 2, 2628 CD Delft
The Netherlands
{r.h.m.huijsmans, b.duz}@tudelft.nl

Bogdan Iwanowski
Rik Wemmenhove
FORCE Technology Norway AS
Claude Monets alle 5
1338 Sandvika, Norway
{boi, riw}@force.no

Mart J.A. Borsboom
Peter R. Wellens
Deltares
P.O. Box 177, 2600 MH Delft
The Netherlands
{mart.borsboom, peter.wellens}@deltares.nl

Henri J.L. van der Heiden
Peter van der Plas
Institute for Mathematics and Computer Science
University of Groningen
P.O. Box 407, 9700 AK Groningen
The Netherlands
{h.j.l.van.der.heiden, p.van.der.plas}@rug.nl

ABSTRACT

Hydrodynamic wave loading on structures plays an important role in areas such as coastal protection, harbor design and offshore constructions (FPSO's, mooring), and there is a need for its prediction up to a detailed level (max./min. pressures, duration of pressure peaks, shear stresses, etc.). In close cooperation with industry, long-year joint-industry projects are carried out to develop a numerical simulation method: the CFD method ComFLOW. The two major application areas are the prediction of extreme wave forces on offshore platforms and offloading vessels, and the prediction of impact forces on coastal protection structures. The paper will present a short overview of the method, some recent results and future plans.

*Address all correspondence to this author.

INTRODUCTION

Extreme waves and their impact loading on fixed and floating structures, like production and offloading platforms, coastal protection systems and offshore wind farms, have long been subjects that could only be studied with experimental methods. They can be a serious threat to the land behind the dikes and its inhabitants. Also the safety and operability of offshore vessels and the well-being of their crews are jeopardized. The 2004/2005 hurricanes Ivan, Katrina and Rita in the Gulf of Mexico have dramatically refocussed attention to extreme waves and their consequences for coastal defense systems and offshore structures [1, 2] (see Fig. 1). These hurricanes created huge devastations both on land and at sea, causing many casualties and huge economic damage. Thus, a better understanding of the consequences of these forces of nature is urgently needed.

FIGURE 1. *Left:* AFTER HURRICANE IVAN, THE ENSCO PLATFORM WAS FOUND 40 MILES FROM WHERE IT WAS ANCHORED. *Right:* DAMAGED BREAKWATER.

In particular, (ship-type) offshore units for the production and storage of oil (FPSOs) should be able to survive the most critical environmental conditions occurring as they are unable to flee for approaching storms. This requires an adequate mooring system, but also attention to the potential problem of green water on the deck. The latter can cause large damage to the vessel's sensitive superstructure and equipment, such as the fluid swivels, piping, turret structure, control valves, emergency systems, fire detection/protection systems, and cable trays.

Similar problems can occur due to wave loading on offshore windmills, often gathered together in large wind farms. Extreme hydrodynamic loads may cause severe damage to their support structures, and herewith jeopardize their operability [3].

Also, coastal defense systems (Fig. 1) will have to withstand larger forces of nature because of changing conditions (sea level rise, increasing storm surges due to climate changes), therefore it continues to be necessary to improve and update the predictions of the wave climate near the coast [4]. Besides the energy of the wind waves generated at sea, the hydrodynamic load of coastal structures is also determined by foreshore processes like shoaling, breaking and refraction.

Experiment versus theoretical model

At present, the addressed phenomena are mainly studied experimentally, and the results are used for the formulation of design rules. The experimental data, however, is far from complete, because of the large number of physical parameters that is involved and the costs of experimental research. Thus there is a growing need for a numerical simulation tool capable of predicting in detail the hydrodynamic load due to waves and currents, and its effect at and near structures (see e.g. [5–7]). A numerical model has the advantage that simulations can quickly be adapted to small changes in geometry or conditions, that scaling effects can be avoided, and that detailed insight in the hydrodynamic processes can be obtained. The instantaneous availability of a numerical model is another important advantage.

The tools currently available are hardly capable of predicting such events to an acceptable level of accuracy, as these tools largely depend on the application of simple models based on e.g. linear potential flow theory or shallow-water theory [8]. As a consequence the applications were restricted to mildly non-linear flow phenomena. In contrast, the physical phenomena accompanying extreme events are both highly non-linear and highly dispersive due to the occurring wave steepness, and require new methods as a basis for the prediction of the water flow and it induced hydrodynamic loads.

The mathematical model for complex water flow dates from the first half of the 19th century already and is known as the Navier–Stokes equations. However, it is only for about a decade that these field equations can be solved for large-scale complex free-surface flow problems, thanks to novel numerical algorithms and the increase in computer power. This recent step to fully non-linear flow modelling based on the Navier–Stokes equations has been stimulated by pioneering work carried out at Los Alamos National Labs, e.g. [9, 10].

As always, many roads lead to Rome and the literature shows a variety of approaches; we mention some developments. At Principia (France) a simulation tool (called EOLE) based on the VOF technique is under development, at INSEAN (Italy) in cooperation with the University of Santa Barbara (USA) programs based on smoothed particle hydrodynamics are being designed, e.g. [11], whereas at ECN (France) a non-linear potential flow solver is coupled with a high-Reynolds Navier–Stokes solver [12]. In the coastal engineering area, next to Deltares' efforts (see below), the research carried out at Ghent University (Belgium) should be mentioned, e.g. [13]. For an overview of Navier–Stokes methods for free-surface flow we refer to [14].

The Dutch contribution

In this paper we will describe a (mainly) Dutch contribution to hydrodynamic simulation models based on the Navier–Stokes equations, in particular featuring the COMFLOW method. In close cooperation between the University of Groningen, the Technical University of Delft, MARIN, FORCE Technology and Deltares, it is especially designed to simulate the steep waves near, at and around fixed and floating structures, e.g. offshore platforms and coastal breakwaters. The development is embedded in Joint Industry Projects (JIP), led by Marin, and supported by several offshore-related companies (oil companies, ship yards, classification societies, engineering firms). Several of these companies are using COMFLOW for their regular activities (design, consultancy).

Over the years several publications (by both developers and users) have appeared on separate aspects of ComFLOW. In the present paper, we give an overview of all major ingredients of the method; also plans for the future will be discussed. First the history of the ComFLOW developments is recalled, and the main lines of the method will be sketched. Some major improvements will be presented, like the local height function, gravity-consistent density averaging and non-reflecting boundary conditions. These are demonstrated on several validated examples of extreme wave loading. Thereafter, we will present some new developments to be carried out in the near future, like turbulence modeling for coarse grids by means of regularization models, the treatment of non-conforming boundaries (cut cells), and local grid refinement.

HISTORY

Spacecraft dynamics The first steps towards the development of a free-surface simulation method were made in the late 1970s, when the National Aerospace Laboratory NLR was studying the influence of liquid propellant (or other liquids) on-board spacecraft. This resulted in the Savof code [15]. Later, in the mid-1990s the development of free-surface simulation methods was continued at RUG to create a method for fully three-dimensional flow; see e.g. [16]. This included the design of full dynamical coupling between liquid motion and spacecraft motion [17,18]. One of the highlights, early 2005, was the space flight of the experiment satellite Sloshsat FLEVO: basically a tank partially filled with water. With this NLR-built spacecraft a large series of experiments involving liquid sloshing under micro-gravity were performed [18,19].

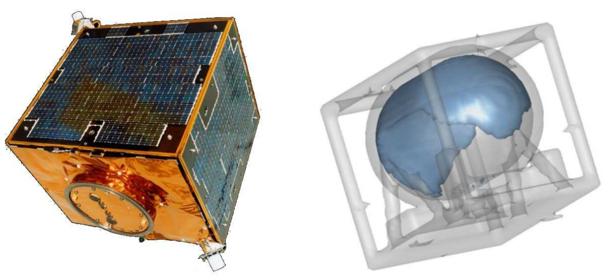

FIGURE 2. Sloshsat FLEVO *(left)*, SIMULATED SNAPSHOT *(right)*.

Under micro-gravity, capillary effects at the free liquid surface are dominantly present. Since these effects are proportional to the curvature of the liquid surface, high accuracy requirements are put on the description and displacement of the free surface. To meet these requirements, the original VOF treatment for reconstruction and movement of the free surface (with its considerable amount of 'flotsam and jetsam'; see Fig. 7) has been extensively redesigned. The use of a local height function was found to be crucial [16], also in the maritime applications that would follow (see below).

Maritime applications In the late 1990ies, MARIN learned of the micro-gravity applications at RUG with their dynamic free-surface motion. The idea came up to test this approach for violent wave motion at sea. The cooperation between RUG and MARIN started in 1998 with an MSc project concerning green water loading, where a fixed bow was subjected to a simplified green water event [20]. This project was later continued as a MARIN-funded PhD project, where the physics was extended to include the coupled dynamics between the incoming water and the vessel motion [21].

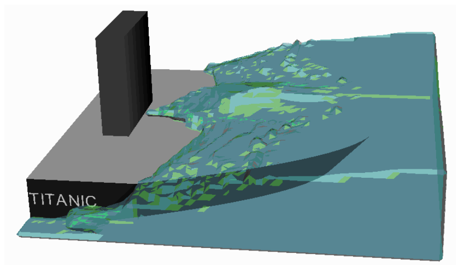

FIGURE 3. GREEN WATER LOADING.

From 2000 onward several joint industry projects have been defined: SafeFLOW funded by the EU, and ComFLOW-2 and ComFLOW-3 funded by STW. These projects were supported by a world-wide consortium of offshore-related companies. Focal point was (and is) on hydrodynamic wave loading. In the SafeFLOW-JIP, as a model test case for green water research, the first (one-phase) dambreak results were produced, as shown in Fig. 4; details can be found in [18, 22, 23]. Some applications have been described in [24–26].

As a follow-up, in 2004 the ComFLOW-2 JIP was started, in which the functionality of the simulation method was enlarged. We will present the most relevant developments below. Since then, several industrial participants have actively been using the ComFLOW program. In 2009/2010 the ComFLOW-3 JIP was started; we will also describe the most important issues in that project.

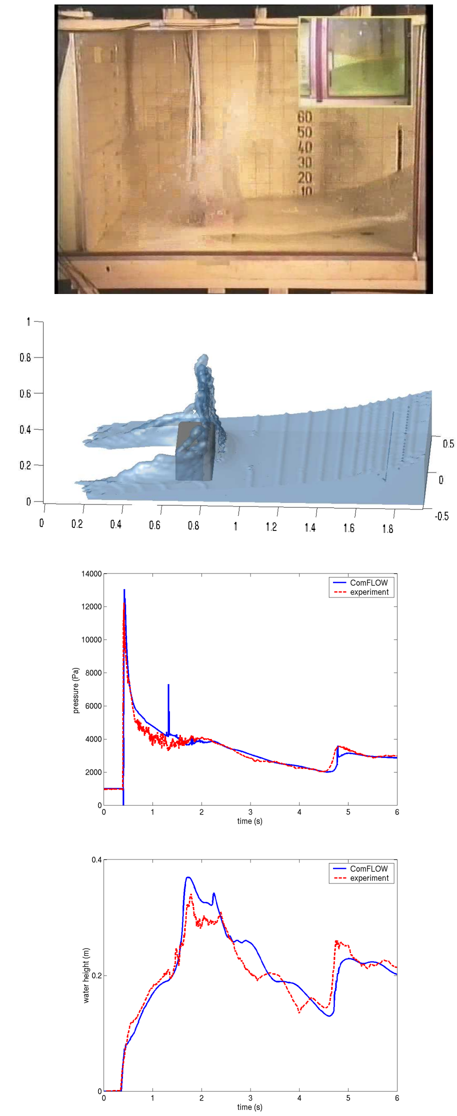

FIGURE 4. WATER IMPACT AGAINST AN OBSTACLE IN A DAMBREAK SETTING: EXPERIMENT VERSUS COMPUTATION. THE LOWER FIGURES SHOW A COMPARISON OF PRESSURE AT THE FRONT OF THE BOX, AND WATER HEIGHT SLIGHTLY BEFORE.

Coastal applications It was already in the mid 1980ies, while the SAVOF simulation method was being developed at NLR, that the first contacts were made with Deltares (then WL Delft Hydraulics). They extended the (then still two-dimensional) SAVOF method [15] to study wave impact against coastal protection systems (dikes, etc.): this resulted in the SKYLLA method [27, 28]. Fig. 5 shows an example of a SKYLLA simulation. Deltares' interest is in reliably and efficiently simulating wave impacts on coastal structures including the damage this leads to. Because the limitation to two dimensions is significant in many applications, in 2006 Deltares decided to join the COMFLOW development.

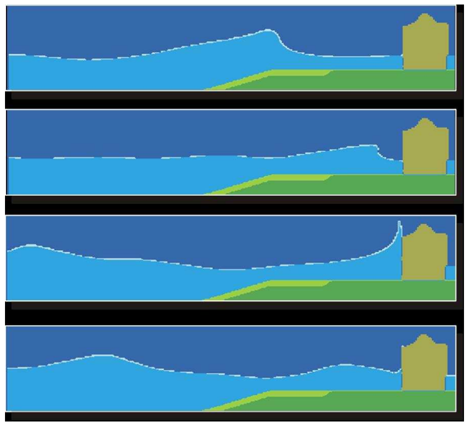

FIGURE 5. SKYLLA CALCULATION OF WAVE RUN-UP AGAINST A BREAKWATER.

DESCRIPTION OF COMFLOW

Flow model

The flow model inside COMFLOW is based on the Navier–Stokes equations. It considers interaction between water and compressible air, i.e. it is a two-phase flow model. This makes it possible to study for example the cushioning effect of entrapped air during wave impacts related to sloshing in LNG tanks (see below).

The two phases are described as one aggregated fluid with varying properties (depending on the mixture ratio), such that the flow can be described by one continuity equation and one momentum equation. In conservation form, for a volume V with boundary S, the flow equations read

$$\int_V \frac{\partial \rho}{\partial t}\,\mathrm{d}V + \oint_S \rho\mathbf{u}\cdot\mathbf{n}\,\mathrm{d}S = 0 \qquad (1)$$

(with $\mathbf{u} = (u,v,w)^T$ the velocity and ρ the density) and

$$\int_V \frac{\partial(\rho\mathbf{u})}{\partial t}\,\mathrm{d}V + \oint_S (\rho\mathbf{u}\cdot\mathbf{n})\mathbf{u}\,\mathrm{d}S + \oint_S p\mathbf{n}\,\mathrm{d}S - \oint_S (\mu(\nabla\mathbf{u}+\nabla\mathbf{u}^T) - \tfrac{2}{3}\mu\nabla\cdot\mathbf{u})\cdot\mathbf{n}\,\mathrm{d}S - \int_V \rho\mathbf{F}\,\mathrm{d}V = 0. \qquad (2)$$

A polytropic equation of state relates pressure p and density ρ.

Generating and absorbing boundary conditions

The computational boundaries of the domain can be a source of non-physical wave reflections, which should be minimized. These inflow and outflow boundaries should be transparent for the incident waves and absorb the outgoing waves at the same time. For monochromatic waves, where the wave number is known beforehand, many approaches are available in the literature, mostly based on some form of the Sommerfeld condition [29]. However, for dispersive waves with a broad spectrum of wave numbers, adequate non-reflecting conditions are much harder to formulate. Thus, in the project, a new type of generating and absorbing boundary conditions (GABC) is being developed which suppresses the wave reflections for a broad range of wavenumbers. This functionality makes it possible to place the boundaries close to the object, thereby reducing the computational time considerably.

Like many other non-reflecting boundary conditions, the GABC is based on a Sommerfeld-type condition: $\phi_t + c(k)\phi_x = 0$. This works fine when the propagation speed c of the waves (a function of the wavenumber k) is known. However, in deeper water dispersion sets in, and any wave becomes a complex mixture of several simple wave components, each propagating at its own dispersive wave speed. Thus a novel idea has been introduced to determine 'local' wave velocities. The first observation is that a linear vertical wave profile can be written as $\phi = C_1 e^{kz} + C_2 e^{-kz}$. The next step is to recognize that $k^2 = \phi_{zz}/\phi$. The dispersion relation gives the wave velocity c as a function of the wavenumber k, and the latter observation makes it a function of ϕ_{zz}/ϕ. Herewith the Sommerfeld condition becomes

$$\phi_t + c\,(\sqrt{\phi_{zz}/\phi})\,\phi_x = 0. \qquad (3)$$

The linear dispersion relation $c = \sqrt{(g/k)\tanh(kh)}$ is approximated by a Padé approximation

$$c \approx \sqrt{gh}\,\frac{a_0 + a_1(kh)^2}{1 + b_1(kh)^2}, \qquad (4)$$

where the coefficients a_0, a_1 and b_1 are chosen to minimize reflections over a user-specified wavenumber range. Substituting (4) into (3) results in

$$\left(1 + b_1 h^2 \frac{\partial^2}{\partial z^2}\right)\frac{\partial \phi}{\partial t} + \sqrt{gh}\left(a_0 + a_1 h^2 \frac{\partial^2}{\partial z^2}\right)\frac{\partial \phi}{\partial x} = 0. \qquad (5)$$

Next, by realizing that $(\partial/\partial x)\phi = u$, and $(\partial/\partial t)\phi = -(p/\rho) - gz$, (5) can be written in terms of the primary variables u and p. Finally, with the momentum equation (2), the contribution from u can be written in terms of p. Thus, ultimately one turns up with a boundary condition for the pressure p. Because of the second derivative $\partial^2/\partial z^2$, it has a larger stencil than usual, which has implications for the pressure Poisson solver.

Thusfar, the GABC has been tested for low-amplitude waves. Extension to more extreme waves is foreseen in the follow-up project. For details we refer to the forthcoming PhD thesis of Wellens at TU Delft, and to [30].

Basic numerics

ComFLOW belongs to the class of Volume-of-Fluid (VOF) methods. It solves the Navier–Stokes equations on a staggered grid [31], where for each computational cell the VOF-function indicates which fraction of it is filled with liquid [9]. In principle, this method is strictly mass conserving (but see below), which is highly relevant in our applications. This is in contrast to the level-set method [32], which is popular in other types of applications where mass conservation is not an important issue.

The computational grid is chosen rectangular; the simplicity of the grid gives an easy geometric framework in which the position and slope of the surface can be accurately described. On unstructured grids always some kind of smearing of the surface is necessary to describe the position of the free liquid surface. This creates a 'spongy' surface, which will reduce peak pressures during impact. In our application (prediction of wave forces) this is not acceptable.

Rectangular (Cartesian) grids can be generated easily, and allow simpler data structures enabling easier development. Further, bodies can move freely through the grid. The body description is also of VOF-type, which keeps the shape of the body crisp. Note that an immersed boundary treatment [33] would smear out the body, which again will suppress pressure peaks. Also, in principle collisions of bodies are allowed, without the grids being squeezed in between; the latter would have been the case when boundary conforming grids were used. A drawback may be that due to the non-boundary conforming character, the resolution of viscous boundary layers requires additional attention. However, sufficiently powerful 'cut-cell' techniques [34, 35] are available.

Algorithmic highlights

Application of, seemingly well-established, numerical techniques to realistic flow problems often ruthlessly reveals any flaw in the numerical treatment. In the development of ComFLOW the situation was not much different. Thus, at several places inside the algorithm, numerical

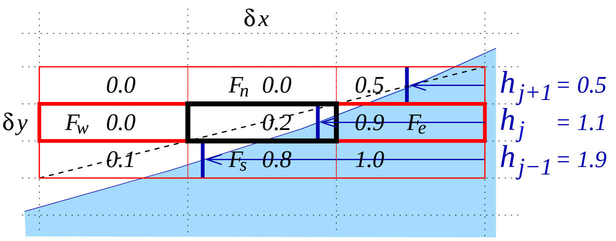

FIGURE 6. LOCAL HEIGHT FUNCTION; HERE THE ORIENTATION IS VERTICAL.

building stones had to be redesigned at a quite fundamental level. We mention some of these innovative 'highlights'.

Local height function As mentioned above, the original VOF method does suffer from 'flotsam and jetsam', i.e. loose droplets that separate from the liquid surface as a numerical artifact of the liquid displacement algorithm. In the calculations this show as intensive spray; see for instance Fig. 7. Also this leads to (sometimes serious) numerical loss of mass, although in theory VOF is fully mass conserving. To suppress the droplets, the basic VOF displacement algorithm was adapted. Hereto, a local height function was introduced [16, 22], as one would naturally use for a more or less horizontal water surface.

In broad lines, locally the surface is described by a height function in a suitably chosen direction. In Fig. 6 this is the vertical direction (because the slope of the surface is larger than the slope of the cell diagonal). Perpendicular to this direction, the VOF-values are added so as to form an, in this case horizontal, height function. For the middle one of the three rows in Fig. 6, the flux in and out of this row is calculated, providing a new value for the height function. Finally, the new VOF-values for each of the cells in the middle (thick red-encadered) row can be reconstructed. In this way the liquid keeps together, without droplets being split of numerically. The positive effect of this procedure can be seen in Fig. 7. This approach has also appeared elsewhere in the literature, e.g. [36, 37].

Gravity-consistent discretization Two-phase calculations have to deal with a discrete treatment of the density, which varies significantly between the two phases. In grid points near the free surface some form of density averaging has to be used. These averaging formulas may lead to (sometimes large) unphysical velocities, often called spurious or parasitic currents [37–39]. Their origin can be understood by considering an equilibrium configuration, with zero velocity, where a balance exists between body force

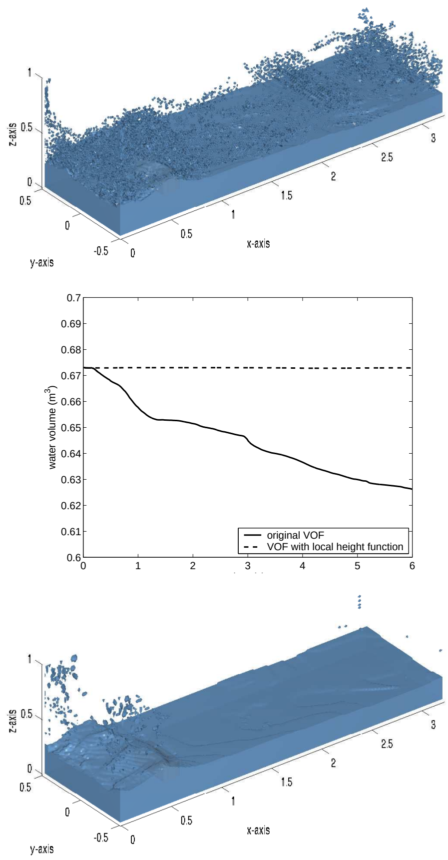

FIGURE 7. 'FLOTSAM AND JETSAM' AFTER A DAMBREAK CALCULATION WITHOUT *(top)* AND WITH *(bottom)* A LOCAL HEIGHT FUNCTION. THE MIDDLE FIGURE SHOWS THE MASS LOSS.

($\mathbf{F}$) and pressure gradient, i.e.

$$\frac{1}{\rho}\,\text{grad}\; p = \mathbf{F} \quad \Leftrightarrow \quad \text{grad}\; p = \rho\mathbf{F} \quad \Leftrightarrow \quad 0 = \text{curl}(\rho\mathbf{F}). \quad (6)$$

For equilibrium conditions, the right-most equality also should hold in the discrete approximation, otherwise the u-dependent terms in the momentum equation no longer vanish and spurious velocities are generated. Herewith, it puts a requirement on the way in which ρ is averaged. When $\mathbf{F}$ denotes gravity, averaging formulas for ρ that satisfy this requirement are called gravity consistent.

The staggering of the grid is responsible for the problems. As indicated in Fig. 8, densities are defined in cell centers (i and $i+1$), where they represent a cell average. However, the momentum equation in $i+1/2$ also requires a density value, which has to be obtained via some interpolation procedure. A logical choice would be to weight both

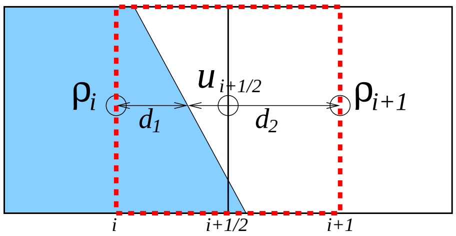

FIGURE 8. DENSITY AVERAGING. THE RED-DASHED REGION IS THE MOMENTUM CONTROL VOLUME.

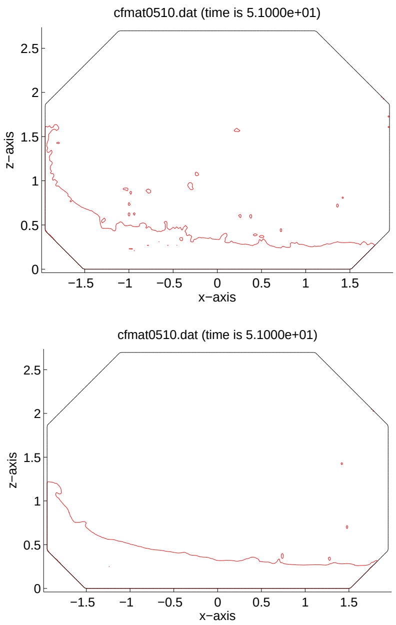

FIGURE 9. THE EFFECT OF THE GRAVITY-CONSISTENT DENSITY AVERAGING IN A SLOSHING SIMULATION: THE 'USUAL' APPROACH *(top)* VERSUS THE NEWLY DEVELOPED APPROACH *(bottom)*.

density cell values with the fraction of the cells that is covered by liquid and air, respectively. But this combination turns out not to be gravity consistent. The better combination is to look at only the momentum control volume (the red-dashed region), and weight the neighbouring density values with the fraction that this control volume is split [40]. Thus (for notation referring to Fig. 8)

$$\rho_{i+1/2} = \frac{d_1 \rho_{i+1} + d_2 \rho_i}{d_1 + d_2}. \tag{7}$$

In most of the literature, the spurious currents are damped with some form of diffusion, or with other suppression techniques. However, explicit mentioning of the above requirement (6) that fully prevents these unwanted currents is scarce, although some authors are close, e.g. [37].

RESULTS

Validation

In the ComFLOW-2 project, a well documented and accurate set of validation data has been obtained by means of model tests. We will present some results from two types of experiments:

a) Sloshing in an LNG tank (1:10 scale).
b) Wave run-up on a semi-submersible.

Animations of several ComFLOW applications can be found at the website [41].

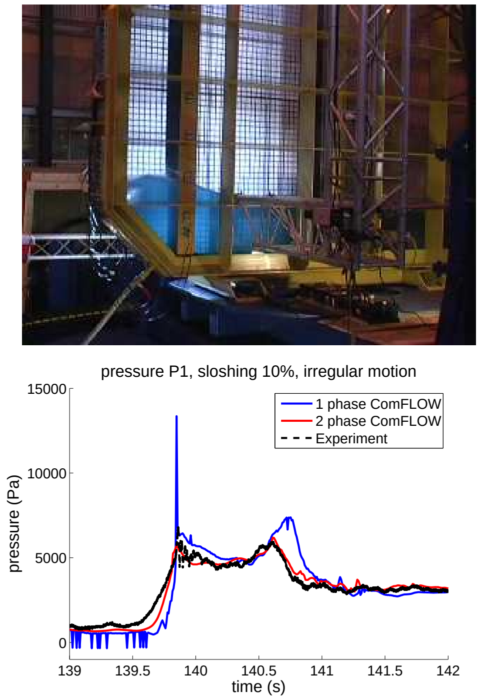

FIGURE 10. SLOSHING: ENTRAPPED AIR IN EXPERIMENT *(top)*; VALIDATION OF PRESSURE SIGNAL *(bottom)*.

Sloshing tests have been carried out for various filling rates, ranging from 10% up to 95%. In the latter case, compressibility effects are clearly visible [18, 42], but also during air entrapment of breaking waves. As an example, Fig. 10 shows an entrapped air bubble during the collapse of an overturning wave against a solid tank wall in an irregular 10% sloshing test. Here the improvement in force (pressure) prediction that is obtained with two-phase modelling over a one-phase model is clearly visible.

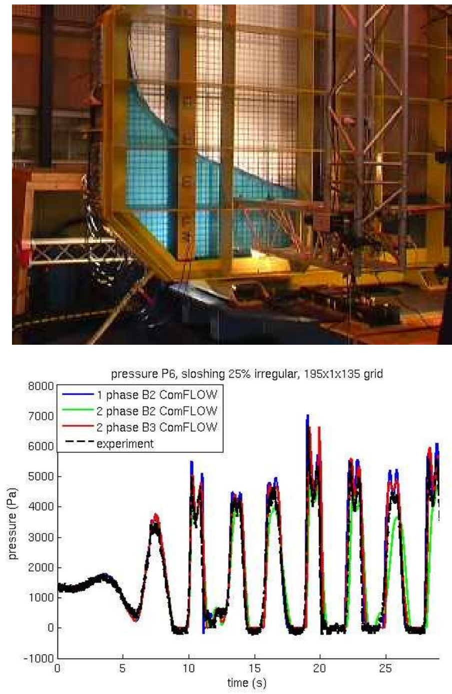

FIGURE 11. SNAPSHOT OF EXPERIMENT *(top)* AND VALIDATION OF PRESSURE FOR 25% SLOSHING TEST *(bottom)*.

An example of a 25% sloshing test is shown in Fig. 11. The experimental data are compared with several modelling options: one-phase, two-phase versus two- or three-point backward upwind discretization (i.e. first or second order accurate). In the two-phase flow calculation, the influence of the air is clearly present due to the numerical (=non-physical) diffusion of a first-order upwind discretization. This is a general tendency we saw in many calculations. With a two-phase flow model, it is necessary to reduce the numerical diffusion, e.g. by using a second-order upwind method. Other presentations of the sloshing tests and their corresponding ComFLOW calculations can be found in [43, 44].

Also, combined experimental and computational studies of the wave run-up against a semi-submersible have been made [45–47]; Fig. 12 gives an impression.

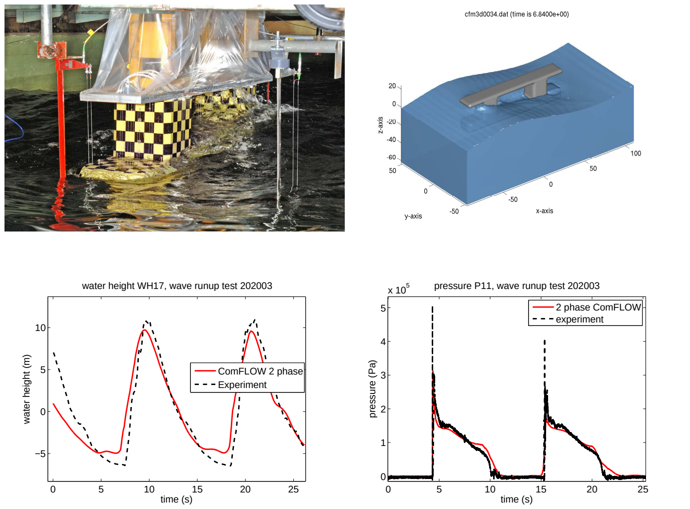

FIGURE 12. VALIDATION OF FORCES ON SEMI-SUBMERSIBLE: WATER HEIGHT *(left)* AND PRESSURE *(right)*

Results by industrial users

Several industrial JIP-participants have actively been using the ComFLOW program. We mention some examples.

Fig. 13 shows a simulation of the Statoil Snorre tension leg platform, which includes dynamic coupling with the anchor lines. Hereto, an interactive numerical coupling between fluid dynamics, floater motions and mooring system has been implemented. It is fully implicit, with subcycling inside each time step [48]. However, it cannot be used in conjunction with (binary) third-party codes for the body motion, as then only information exchange per time step is possible (see below). Other examples of wave effects interacting with body motion are, e.g., a subsea structure lifted through the splash zone [49] and an interactive sloshing tank [50].

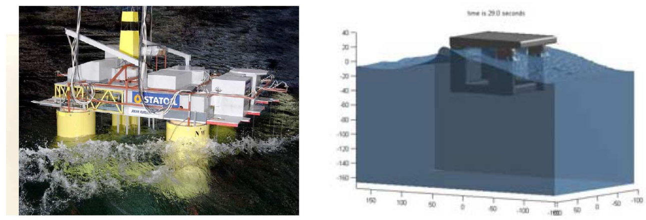

FIGURE 13. INTERACTIVE MOTION OF SNORRE TLP: MODEL TEST AND SNAPSHOT OF SIMULATION (STATOIL-HYDRO [48]).

Another example is the run-up against gravity-based structures studied by Chevron-Texaco [51, 52] and shown in Fig. 14. Furthermore, wave-in-deck studies have been reported by DNV [53, 54], as well as by Force Technology [55].

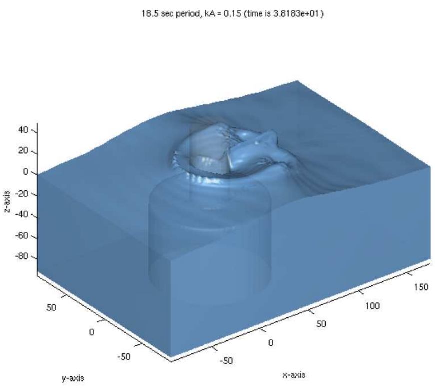

FIGURE 14. WAVE RUN-UP AGAINST VERTICAL COLUMN, SHOWING 'ROOSTERTAIL' (CHEVRON-TEXACO [52]).

FUTURE PLANS

Over the years, the ComFLOW method has evolved into a powerful basic computational tool for pursuing the challenges of violent weather-induced marine and coastal applications. The above examples are still fairly limited in physical contents; a desire exists to extend the functionality of the method. For the recently started ComFLOW-3 project several new features are foreseen; we give an impression below.

In order to reach the above new functionality and improved efficiency, several basic algorithmic and modelling building blocks have to be devised. We present a short discussion.

GABC extensions The newly developed Generating and Absorbing Boundary Condition (GABC) is intended to absorb waves at the inflow and outflow boundaries, and to generate irregular waves at the inflow boundary simultaneously. Currently, the method is working well for low (two-dimensional) waves propagating perpendicular to the boundary. The GABC will be extended to (three-dimensional) short-crested outgoing waves and waves coming from different directions. Hence the GABC (3) will be extended to describe waves from several angles. For the latter a product (chain) of Sommerfeld conditions can be applied; see e.g. [29]. Our first experiences reveal that with two factors a fair range in incidence angles can be covered, whereas at the same time the complexity of the resulting boundary condition, cf. (5), is still manageable. Also, the GABC will be made suitable for more extreme waves, and to include the effects of stationary currents [30].

Combined wave-type and viscous effects Until now, the ComFLOW projects have primarily focussed on extreme wave effects that were highly momentum driven and in which effects of viscosity were of relatively small importance. However, there are several applications in which the effect of viscosity, possibly in combination with wave effects, requires attention. Some examples: wave interaction effects between floaters in side-by-side offloading; sloshing in moonpools; roll damping and bilge keels; scouring and seabed friction. With grids restricted in resolution by CPU limitations, it is important to look for turbulence models that can do a reasonable job on coarse grids. Fortunately, during the last decade much improvement has been made in this field. It was found that the preservation of global flow quantities, like energy or helicity, plays an essential role. Thus energy-preserving discretization methods have been designed for compressible and incompressible flow [56–61] (they are already partly implemented in ComFLOW). The new step is an 'energy-preserving' turbulence model, which does not employ eddy-viscosity, but instead the production of small scales is controlled by the (numerical) treatment of the convective term [62–64]. First experiences allow grids much coarser than before, with hope-giving computational effort.

Improved numerical efficiency Even with the future GABC (to decrease the size of the computational domain) and the new class of turbulence models (to lower the need for grid resolution), computational times will remain considerable. Therefore, further improvements are sought along the numerical side, like trying to further decrease the number of required grid points. Especially pressure loads or phenomena like local wave breaking require a grid resolution that is higher than in the more 'quiet' parts of the flow domain. Thus local grid refinement will be pursued (for the moment user-defined, as it reasonable to predict where the interesting flow phenomena will occur). Much literature already exists from which we can borrow [65–67]. New is the combination of grid refinement with a free surface. The displacement algorithm for the free surface will have to be adjusted to the local changes in grid structure. Ideas will be developed in the new project. Other actions are looking for faster sparse Poisson solvers (remember that the GABC induces an irregular matrix stencil near the boundaries), and, of course, parallelization to facilitate faster simulations on simple multi-core PC's.

Interactive body motion Above, a first example of a calculation of interactive body motion has been shown (the Snorre TLP). This was achieved by strongly intertwining the mechanical motion algorithm with the flow solver. In general, such a strong connection, where both sets of dynamical equations are solved simultaneously, is not feasible (e.g. when one of the solvers is a 'black box'). Yet, when a large range in mass and inertia ratios has to tackled, a strong coupling is required. Similar problems occur in the aerodynamic world, and here a good alternative has been developed [68–70]: the quasi-simultaneous approach. We plan to apply it here, by adding a simple approximation of the mechanical model to the flow equations inside ComFLOW. This simple approximation only helps to enhance the convergence of the iterations, and can be considered as a preconditioner. The weak coupling with the black-box mechanics then 'only' has to cope with the difference between the 'exact' dynamical model and its approximation. This provides unconditional stability of the coupling. The coupling approach can be used for small as well as large body motion; the quality of the results is only determined by the 'exact' external dynamical model used.

SUMMARY

An overview has been presented of the ComFLOW simulation method, designed to study hydrodynamic forces on offshore platforms and coastal constructions. Its development history has been sketched, and the models and methods behind ComFLOW have been highlighted. An impression has been given of its current capabilities, with several examples from industry. Finally, the future steps in its development have been indicated.

ACKNOWLEDGMENT

The research is supported by the Dutch Technology Foundation STW, applied science division of NWO and the technology program of the Ministry of Economic Affairs in The Netherlands (contracts GWI.6433 and 10475).

REFERENCES

[1] Wisch, D. J., and Ward, E. G., 2007. "Offshore standards - the impact of hurricanes Ivan/Katrina/Rita". In Proc. 26th OMAE Conf., San Diego, USA.

[2] Forristal, G., 2007. "Wave crest height and deck damage in hurricanes Ivan, Katrina and Rita". In Proc. OTC, Houston.

[3] Haver, S., 2000. "Evidences of the existence of freak waves". In Rogue Waves 2000, Proc. of Int. Workshop, Ifremer, ed., pp. 129–140.

[4] Groeneweg, J., van Ledden, M., and Zijlema, M., 2006. "Wave transformation in front of the Dutch coast". In Proc. 30th Int. Conf. Coastal Engineering (Vol. 1), J. M. Smith, ed., World Scientific, pp. 552–564.

[5] Faltinsen, O. M., 1999. *Sea Loads on Ships and Offshore Structures*. Cambridge University Press.

[6] Buchner, B., 2002. "Green water on ship-type offshore structures". PhD Thesis, University of Delft, Delft, The Netherlands, November.

[7] Molin, B., and Ferziger, J., 2003. "Hydrodynamique des structures offshore". *Appl. Mech. Rev.,* ***56***, p. B29.

[8] Tsai, W., and Yue, D. K. P., 1996. "Computation of nonlinear free-surface flows". *Ann. Rev. Fluid Mech.,* ***28***, pp. 249–278.

[9] Hirt, C. W., and Nichols, B. D., 1981. "Volume of fluid (VOF) method for the dynamics of free boundaries". *J. Comput. Phys.,* ***39***, pp. 201–25.

[10] Rider, W. J., and Kothe, D. B., 1998. "Reconstructing volume tracking". *J. Comput. Phys.,* ***141***, pp. 112–152.

[11] Greco, M., Faltinsen, O. M., and Landrini, M., 2002. "Numerical simulation of heavy water shipping". In Proc. 17th Workshop on Water Waves and Floating Bodies.

[12] Ferrant, P., 1997. "Simulation of strongly nonlinear wave generation and wave-body interactions using a 3D MEL model". In Proc. 21st Symp. on Naval Hydrodynamics, pp. 93–109.

[13] Li, T., Troch, P., and Rouck, J. D., 2004. "Wave overtopping over a sea dike". *J. Comput. Phys.,* ***198***, pp. 686–726.

[14] Osher, S. J., and Tryggvason (eds.), G., 2001. "Special issue on 'Computational methods for multiphase flows"'. *J. Comput. Phys.,* ***169***, pp. 249–759.

[15] Veldman, A. E. P., and Vogels, M. E. S., 1984. "Axisymmetric liquid sloshing under low gravity conditions". *Acta Astronautica,* ***11***, pp. 641–649.

[16] Gerrits, J., 2001. "Dynamics of liquid-filled spacecraft". PhD thesis, University of Groningen, The Netherlands. URL: `dissertations.ub.rug.nl/faculties/science/2001/j.gerrits`.

[17] Gerrits, J., and Veldman, A. E. P., 2003. "Dynamics of liquid-filled spacecraft". *J. Eng. Math.,* ***45***, pp. 21–38.

[18] Veldman, A. E. P., Gerrits, J., Luppes, R., Helder, J. A., and Vreeburg, J. P. B., 2007. "The numerical simulation of liquid sloshing on board spacecraft". *J. Comput. Phys.,* ***224***, pp. 82–99.

[19] Vreeburg, J. P. B., 2008. "Sloshsat spacecraft calibration at stationary spin rates". *J. Spacecraft Rockets,* ***45***(1), pp. 65–75.

[20] Fekken, G., Veldman, A. E. P., and Buchner, B., 1999. "Simulation of green-water loading using the Navier-Stokes equations". In Proc. 7th Int. Conf. Numer. Ship Hydrodyn., J. Piquet, ed., pp. 6.3–1–6.3–12.

[21] Fekken, G., 2004. "Numerical simulation of free-surface flow with moving objects". PhD Thesis, University of Groningen, The Netherlands.

URL: `dissertations.ub.rug.nl/faculties/science/2004/g.fekken`.

[22] Kleefsman, K. M. T., 2005. "Water impact loading on offshore structures - a numerical study". PhD thesis, University of Groningen, The Netherlands. URL: `dissertations.ub.rug.nl/faculties/science/2005/k.m.t.kleefsman`.

[23] Kleefsman, K. M. T., Fekken, G., Veldman, A. E. P., Iwanowski, B., and Buchner, B., 2005. "A Volume-of-Fluid based simulation method for wave impact problems". *J. Comput. Phys.,* ***206***, pp. 363–393.

[24] Loots, E., Buchner, B., Pastoor, W., and Tveitnes, T., 2004. "The numerical solution of LNG sloshing with an improved Volume-of-Fluid method". In Proc. 23rd Int. Conf. Offshore Mech. Arctic Eng. Paper OMAE2004-51085.

[25] Kleefsman, K. M. T., Loots, G. E., Veldman, A. E. P., Buchner, B., Bunnik, T., and Falkenberg, E., 2005. "The numerical solution of green water loading including vessel motions and the incoming wave field". In Proc. 24th Int. Conf. Offshore Mech. Arctic Eng. Paper OMAE2005-67448.

[26] Iwanowski, B., and Falkenberg, E., 2005. "Investigation of fluid flow kinematics due to an incident regular wave in presence of a large offshore gravity based platform". In Computational Methods in Marine Engineering, P. Bergan, et al., ed., CIMNE, Barcelona, pp. 313–322.

[27] van Gent, M. R. A., 1994. "The modelling of wave action on and in coastal structures". *Coastal Engng,* ***22***, pp. 311–339.

[28] Doorn, N., and van Gent, M. R. A., 2003. "Pressures by breaking waves on a slope computed with a VOF model". In Proc. Int. Conf. Coastal Structures.

[29] Givoli, D., 1991. "Non-reflecting boundary conditions". *J. Comput. Phys.,* ***94***, pp. 1–29.

[30] Duz, B., Huijsmans, R., Wellens, P. R., Borsboom, M., and Veldman, A. E. P., 2011. "Towards a general-purpose open boundary condition for wave simulations". In Proc. 30th Conf. on Ocean, Offshore and Arctic Engineering OMAE2011. Paper OMAE2011-49979.

[31] Harlow, F. H., and Welch, J. E., 1965. "Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface". *Phys. Fluids,* ***8***, pp. 2182–2189.

[32] Sethian, J. A., 1996. *Level Set Methods: Evolving Interfaces in Geometry, Fluid Mechanics, Computer Vision and Materials Science*. Cambridge University Press.

[33] Peskin, C. S., 2002. "The immersed boundary method". *Acta Numerica,* ***11***, pp. 479–517.

[34] Dröge, M., and Verstappen, R., 2005. "A new symmetry-preserving Cartesian-grid method for computing flow past arbitrarily shaped objects". *Int. J. Numer. Meth. Fluids,* ***47***, pp. 979–985.

[35] Cheny, Y., and Botella, O., 2010. "The LS-STAG method; a new immersed boundary/level-set method for the computation of incompressible viscous flows in complex moving geometries with good conservation properties". *J. Comput. Phys.,* ***229***, pp. 1043–1076.

[36] Afkhami, S., and Bussmann, M., 2004. "Height functions-based contact angles for VOF simulations of contact line phenomena". *Int. J. Num. Anal. Mod.,* ***1***, pp. 1–18.

[37] Francois, M. M., Cummins, S. J., Dendy, E. D., Kothe, D. B., Sicilian, J. M., and Williams, M. W., 2006. "A balanced-force algorithm for continuous and sharp interfacial surface tension models within a volume tracking framework". *J. Comput. Phys.,* ***213***, pp. 141–173.

[38] Lorstadt, D., Francois, M. M., Shyy, W., and Fuchs, L., 2004. "Assessment of volume of fluid and immersed boundary methods for droplet computations". *Int. J. Numer. Meth. Fluids,* ***46***, pp. 109–125.

[39] Harvie, D. J. E., Davidson, M. R., and Rudman, M., 2006. "An analysis of parasitic current generation in Volume of Fluid simulations". *Appl. Math. Model.,* ***30***, pp. 1056–1066.

[40] Wemmenhove, R., 2008. "Numerical simulation of two-phase flow in offshore environments". PhD Thesis, University of Groningen, The Netherlands. URL: `dissertations.ub.rug.nl/faculties/science/2008/r.wemmenhove`.

[41] ComFLOW website. URL `www.math.rug.nl/~veldman/comflow/comflow.html`.

[42] Wemmenhove, R., Luppes, R., Veldman, A. E. P., and Bunnik, T., 2007. "Numerical simulation of sloshing in LNG tanks with a compressible two-phase model". In Proc. 26th Int. Conf. Offshore Mech. Arctic Eng. Paper OMAE2007-29294.

[43] Bunnik, T., and Huijsmans, R. H. M., 2007. "Large scale LNG sloshing model tests". In Proc. 17th Int. Offshore and Polar Eng. Conf. ISOPE2007.

[44] Iwanowski, B., Lefranc, M., and Wemmenhove, R., 2010. "Numerical investigation of sloshing in a tank, statistical description of experiments and CFD calculations". In 29th Conf. on Ocean, Offshore and Arctic Engineering OMAE2010. Paper OMAE2010-20335 (10 pages).

[45] Wellens, P. R., Pinkster, J. A., Veldman, A. E. P., and Huijsmans, R. H. M., 2007. "Numerical wave run up calculation on GBS columns". In Proc. ISOPE-2007, J. Chung, ed.

[46] Iwanowski, B., Lefranc, M., and Wemmenhove, R.,

2009. “CFD simulation of wave run-up on a semi-submersible and comparison with experiment”. In Proc. 28th Conf. on Ocean, Offshore and Arctic Eng. OMAE2009. Paper OMAE2009-79052.

[47] Wellens, P. R., Luppes, R., Veldman, A. E. P., and Borsboom, M. J. A., 2009. “CFD simulations of a semi-submersible with absorbing boundary conditions”. In Proc. 28th Conf. on Ocean, Offshore and Arctic Eng. OMAE2009. Paper OMAE2009-79342.

[48] Johannessen, T., Haver, S., Bunnik, T., and Buchner, B., 2006. “Extreme wave effects on deep water TLP’s - Lessons learned from the Snorre A model tests”. In Proc. Deep Offshore Technology 2006.

[49] Buchner, B., Bunnik, T., and Veldman, A. E. P., 2006. “The use of a Volume of Fluid (VOF) method coupled to a time domain motion simulation to calculate the motions of a subsea structure lifted through the splash zone”. In Proc. 25th Int. Conf. Offshore Mech. Arctic Eng. Paper OMAE2006-92447.

[50] Bunnik, T., and Veldman, A., 2010. “Modelling the effect of sloshing on ship motions”. In Proc. 29th Conf. on Ocean, Offshore and Arctic Eng. OMAE2010. Paper OMAE2010-20458 (9 pages).

[51] Bunnik, T., Buchner, B., Veldman, A. E. P., Lee, M.-Y., Finnigan, T., and Moises, A., 2006. “Numerical simulation of complex greenwater and wave loads on offshore structures”. In Offshore Technology Conference OTC2006. Paper OTC 17853.

[52] Danmeier, D. G., Seah, R. K. M., Finnigan, T., Roddler, D., Abault, A., Vache, M., and Imamura, J. T., 2008. “Validation of wave run-up calculation methods for a gravity based structure”. In Proc. 27th Int. Conf. Offshore Mechanics and Arctic Eng., OMAE2008. Paper OMAE2008-57625.

[53] Brodtkorb, B., 2008. “Prediction of wave-in-deck forces on fixed jacket-type structures based on CFD calculations”. In Proc. 27th Int. Conf. Offshore Mechanics and Arctic Eng., OMAE2008. Paper OMAE2008-57346.

[54] Brodtkorb, B., 2008. “Prediction of increased jacket substructure loads due to wave-in-deck diffraction based on CFD calculations”. In Proc. 27th Int. Conf. Offshore Mechanics and Arctic Eng., OMAE2008. Paper OMAE2008-57361.

[55] Iwanowski, B., Gladso, R., and Lefranc, M., 2009. “Wave-in-deck load on a jacket platform, CFD-derived pressures and non-linear structural response”. In Proc. 28th Conf. on Ocean, Offshore and Arctic Eng. OMAE2009. Paper OMAE2009-79053.

[56] Verstappen, R. W. C. P., and Veldman, A. E. P., 1997. “Direct numerical simulation of turbulence at lesser costs”. *J. Eng. Math.,* ***32***, pp. 143–159.

[57] Verstappen, R. W. C. P., and Veldman, A. E. P., 1998. “Spectro-consistent discretization: a challenge to RANS and LES”. *J. Eng. Math.,* ***34***, pp. 163–179.

[58] Vasilyev, O., 2000. “High order difference schemes on non-uniform meshes with good conservation properties”. *J. Comput. Phys.,* ***157***, pp. 746–761.

[59] Verstappen, R. W. C. P., and Veldman, A. E. P., 2003. “Symmetry-preserving discretization of turbulent flow”. *J. Comput. Phys.,* ***187***, pp. 343–368.

[60] Veldman, A. E. P., and Lam, K.-W., 2008. “Symmetry-preserving upwind discretization of convection on non-uniform grids”. *Appl. Num. Math.,* ***58***, pp. 1881–1891.

[61] Kok, J. C., 2009. “A high-order low-dispersion symmetry-preserving finite-volume method for compressible flow on curvilinear grids”. *J. Comput. Phys.,* ***228***, pp. 6811–6832.

[62] Helder, J., and Verstappen, R. W. C. P., 2008. “On restraining convective subgrid-scale production in Burgers’ equation”. *Int. J. Numer. Meth. Fluids,* ***56***, pp. 1289–1295.

[63] Verstappen, R. W. C. P., 2008. “On restraining the production of small scales of motion in a turbulent channel flow”. *Comp. Fluids,* ***37***, pp. 887–897.

[64] Trias, F. X., and Verstappen, R. W. C. P., 2011. “On the construction of discrete filters for symmetry-preserving regularization models”. *Comp. Fluids,* ***40***, pp. 139–148.

[65] Minion, M. L., 1996. “A projection method for locally refined grids”. *J. Comput. Phys.,* ***127***, pp. 158–178.

[66] Iaccarino, G., Kalitzin, G., Moin, P., and Khalighi, B., 2004. Local grid refinement for an immersed boundary RANS solver. AIAA paper 2004–586.

[67] Uzgoren, E., Singh, R., Sim, J., and Shyy, W., 2007. “Computational modeling for multiphase flows with spacecraft application”. *Progress in Aerospace Sciences,* ***43***(4-6), pp. 138–192.

[68] Veldman, A. E. P., 1981. “New, quasi-simultaneous method to calculate interacting boundary layers”. *AIAA J.,* ***19***, pp. 79–85.

[69] Veldman, A. E. P., 2001. “Matched asymptotic expansions and the numerical treatment of viscous-inviscid interaction”. *J. Eng. Math.,* ***39***, pp. 189–206.

[70] Veldman, A. E. P., 2009. “A simple interaction law for viscous-inviscid interaction”. *J. Eng. Math.,* ***65***, pp. 367–383.

