IOP Conf. Series: Materials Science and Engineering **276** (2017) 012009 doi:10.1088/1757-899X/276/1/012009

CFD Analysis in Subsea and Marine Technology

Hrvoje Jasak[1,2]

[1] Wikki Ltd, 459 Southbank House, Black Prince Road, London SE1 7SJ, United Kingdom
[2] Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb, Ivana Lučića 5, 10 000 Zagreb, Croatia

E-mail: h.jasak@wikki.co.uk, hrvoje.jasak@fsb.hr

Abstract. Computational Fluid Dynamics (CFD) is established in design and analysis for a range of industries, but its use in Marine and Naval Hydrodynamics is behind the trend. This can be attributed to the complexity of modelling needs, including presence of free surface, irregular transient flows, fluid-structure coupling and presence of established modelling tools based on potential theory.

In this paper, state-of-the-art of CFD in Naval Hydrodynamics, wave and offshore applications is given, with an update of recent advances, validation and computing requirements for typical simulation cases.

1. Introduction

Since the beginning of the 21st century, Computational Fluid Dynamics (CFD) has become well established across the range of engineering applications. CFD results are well understood, expectations on accuracy in design studies can be satisfied. For example, automotive industry uses CFD to significantly reduce the use of experimental studies in aerodynamics, internal combustion engine design and exhaust gas after-treatment. Indeed, CFD studies are now regularly performed in applications which were never properly analysed experimentally, such as passenger compartment comfort, vehicle soiling and acoustics-related phenomena, where the design primarily relies on CFD results.

Acceptance of CFD in marine engineering, naval hydrodynamics and evaluation of wave loads appears to be lagging behind the trend of other industries. This is partially due to the complexity of free surface flows, but mainly by the fact that the phenomena under consideration imply significantly higher computational requirements. For example, sea-keeping studies of ships in irregular waves or calculation of transfer functions for wave loading on floating structures still cannot be performed at acceptable speed. "Engineering time-scale" in product design requires turn-over time of 8-12 hours for simulations to be practically useful.

An important factor in the acceptance of CFD is the presence of alternative simulation tools. Numerical models based on potential flow theory are widely accepted, fast and validated and have been in use for more than a decade. The potential flow assumption, whole adequate for a part of industrial needs, brings it own limitations, mainly due to the formulation, or failure to account for coupled/non-linear effects. The best example is the evaluation of added resistance in waves, which may provide adequate loading results on the hull, but cannot account for forward speed of the ship.

The challenges for CFD modelling in naval hydrodynamics can be grouped as follows:

IOP Conf. Series: Materials Science and Engineering **276** (2017) 012009 doi:10.1088/1757-899X/276/1/012009

- Handling of discontinuity at the free surface;
- Accurate tracking of free surface position;
- Issues of force evaluation and hydro-mechanical coupling;
- Generation and propagation of surface waves, of various nature: deterministic single- and multi-frequency- wave trains, wave focusing and statistical modelling of realistic irregular sea states;
- Modelling of rigid or flexible floating structures;
- Interaction with global external forces, such as mooring systems and propulsors;
- Modelling of extreme wave events such as freak wave impact or green sea loads;
- Statistics-based models, such as 3-hour-storm or safe return to port simulations and
- Full scale effects.

In what follows, recent advances for each of the listed topics from the author's work shall be presented. The paper will conclude with a readiness assessment of CFD tools for simulation types of varying complexity.

2. CFD Modelling of Free Surface Flows

The two-fluid model for immiscible incompressible free surface flows is well established, consisting of phase continuity, combined momentum equation and global continuity equation.

$$\frac{\partial \alpha}{\partial t} + \nabla\bullet(\mathbf{u}\,\alpha) = 0, \tag{1}$$

$$\frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla\bullet(\rho \mathbf{u}\mathbf{u}) - \nabla\bullet(\mu \nabla \mathbf{u}) = -\nabla p + \rho \mathbf{g}, \tag{2}$$

$$\nabla\bullet\mathbf{u} = 0, \tag{3}$$

where α is the phase fraction, ρ, $\mathbf{u}$ and μ are combined density, velocity and viscosity, by virtue of the immiscibility condition, *e.g.*[1]:

$$\rho = \alpha \rho_1 + (1 - \alpha)\rho_2, \tag{4}$$

$$\mathbf{u} = \alpha \mathbf{u}_1 + (1 - \alpha)\mathbf{u}_2, \tag{5}$$

$$\mu = \alpha \mu_1 + (1 - \alpha)\mu_2. \tag{6}$$

Here, ρ_i, $\mathbf{u}_i$ and μ_i represent phase density, velocity and viscosity, respectively. This model is capable of describing the details of non-linear wave interaction with marine objects.

In practice, the equation set is enhanced by turbulence modelling, dynamic mesh support, coupling with a rigid or elastic motion of solid boundaries, wave generation and absorption *etc.*but sufficiently illustrates the numerical challenges.

3. Discretisation and Numerics for Discontinuous Variables

The Finite Volume Method (FVM) [2] is a method of choice for numerical simulation of fluid flows, due to its combination of strict conservation, simplicity, efficiency and flexibility. Free surface flow modelling brings further challenges, to be addressed below.

3.1. Body Force and Pressure Gradient

The first challenge relates to the handling of body force $\rho\,\mathbf{g}$ in presence of a step change in ρ. In hydrostatics, the body force and pressure gradient need to balanced and discretisation consistency in the momentum equation requires that *the computational stencil for the balancing* *terms needs to match.* In (2) this is clearly not the case, resulting in pressure chequerboarding. To remedy the problem, the pressure variable is redefined:

$$p_d = p - \rho\, \mathbf{g}_\bullet \mathbf{x}, \tag{7}$$

yielding a reformulated balanced momentum source term

$$-\nabla p + \rho \mathbf{g} = -\nabla p_d - (\mathbf{g}_\bullet \mathbf{x}) \nabla \rho, \tag{8}$$

with matching computational stencil for ∇p_d and $\nabla \rho$.

3.2. Interface Jump Conditions

While (8) resolves the pressure chequerboarding problem, it still contains the gradient of the (discontinuous) ρ at the interface. A practical consequence of numerical gradient evaluation for a step-function is the parasitic velocity at the interface. Quetey *et al* [3] take into account the discontinuities at the free surface within the FVM, assuming that the free surface always coincides with the mesh faces. Such an approach requires significant mesh refinement at the free surface. The problem is resolved by the Ghost Fluid Method (GFM) [4] developed by Vukčević, accounting for discontinuities at an arbitrary location in the mesh.

The second-order accurate GFM is derived from the assumption of piece-wise constant phase density, allowing (2) to be rewritten in terms of continuous velocity field $\mathbf{u}$ instead of discontinuous linear momentum $\rho \mathbf{u}$:

$$\frac{\partial \mathbf{u}}{\partial t} + \nabla_\bullet(\mathbf{u}\mathbf{u}) - \nabla_\bullet(\nu \nabla \mathbf{u}) = -\frac{1}{\rho} \nabla p. \tag{9}$$

Presence of the interface is accounted for in *interface jump conditions* with $[\psi] = \psi^- - \psi^+$ denoting the jump of the variable at the interface:

$$[\mathbf{u}] = 0, \tag{10}$$
$$[p] = 0, \tag{11}$$
$$\left[\frac{1}{\rho} \nabla p_d\right] = 0. \tag{12}$$

The jump conditions directly account for discontinuous density and pressure gradient at the interface and are built into the discretisation for pressure gradient and Laplacian. Precise location of the interface is evaluated between the two cell centres straddling the free surface from interface tracking information.

4. Handling of the Free Surface

The primary consideration of modelling of free surface flows is the capturing of position and topology of the free surface. Its position profoundly influences the flow solution to which it is coupled in a non-linear manner. In this section, we shall review some options on free surface handling and implications to the needs of marine CFD.

There are two fundamental approaches. *Surface tracking* deforms the computational mesh at the interface to comply to its current configuration, usually reducing the problem to a moving boundary single-phase flow [5, 6]. *Surface capturing* uses a continuum representation of the free surface and the actual position of the interface is obtained via post-processing. Examples include various forms of marker-in-cell or particle-in-cell simulations, solution of the Volume-of-Fluid (VOF) [1, 7, 8], level-set [9, 10, 11, 12] or similar [13, 14].

4.1. Linearised Free Surface and Surface Tracking

Surface tracking is a straightforward way of dealing with a moving free surface [15], whose position is defined by a double boundary condition:

- Fixed surface pressure, either assuming atmospheric pressure above water level or via pressure continuity in two-phase flows;
- Zero net flux through the surface, defining the surface normal velocity to be equal to the mesh motion flux.

In its simplest form, the free surface boundary condition is linearised in terms of pressure and surface elevation without deforming the mesh, yielding the *linearised free surface model.* Mode complex formulations involving a deforming mesh provide a more accurate description of surface motion with a modest increase in computational cost.

The main advantage of surface tracking is that fact that the free surface problem may be reduced to a single phase, without the need of increased resolution of the computational mesh near the surface. It disadvantages relate to complexity of mesh motion, interaction with solid boundaries of complex shape and most importantly its inability to efficiently handle changes in surface topology such as breaking waves.

4.2. Surface Capturing Models

Surface capturing models use a continuum representation of the free surface, via a volume fraction or a general colouring function. They can be divided into three major groups, illustrated in Fig. 1:

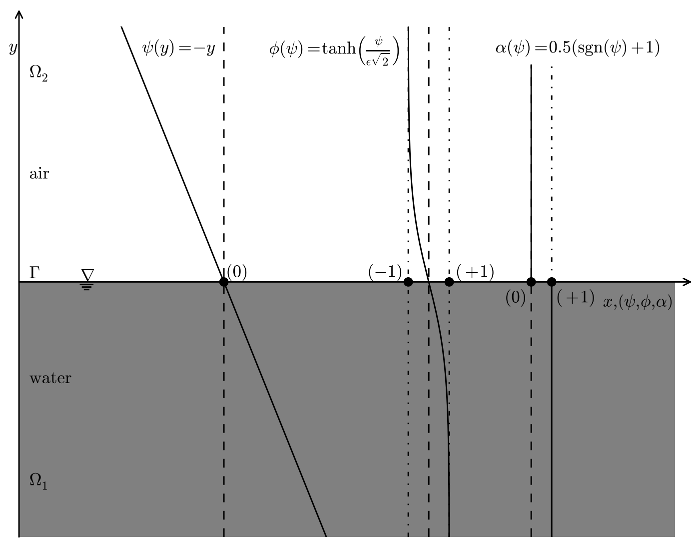

Figure 1. Interface capturing schemes: Level Set, Phase Field and Volume of Fluid.

- Volume of Fluid (VOF) methods (1), [1] is simple to understand and strictly satisfies the phase conservative condition but suffers from the need to advect the step-function in α without unboundedness or smearing;

- Seeking a "smoother" solution variable, Level Set (LS) methods [9] solve for the signed distance to the interface. While the solution no longer contains a step, LS does not guarantee mass conservation either, which is a major drawback. Furthermore, the discretisation error in the transport appears as a loss of unit gradient for the distance function; as a result, the advection step is usually followed by re-initialisation. This can be done by direct calculation of distance [16] in a narrow band, or by solving an additional Eikonal equation to propagate the unit gradient constraint [17]. The best methods use implicit re-initialisation [12] or combine mass conservation properties of VOF and smoothness of LS in some way [11];
- Phase Field (PF) methods are developed on a basis that there exists a transport equation for self-similar advection of a tanh profile [18]. With appropriate mathematical reformulation [12] it is possible to cast the PF equation into the LS form, treating all terms fully implicitly within the FV framework. This effectively combines the advection and re-initialisation step of LS into one.

Based on extensive experience with all of the above, we typically choose VOF for problems where mass conservation is crucial [19]. For solution decomposition methods, it is preferable to choose an unbounded variable and LS methods are preferred [12, 20].

4.2.1. Volume-of-Fluid and Sharp Interfaces While the VOF equation brings advantages such as clear variable bounds and strict phase conservation, its weak point is the shape of the advected profile. For good performance and accurate surface-to-surface interaction it is necessary to preserve the sharpness of the step profile denoting the free surface position as accurately as possible. Various forms of interface compression are used in practice, either based on numerical artefacts of convection schemes [1] or quasi-physical compression [21].

4.2.2. Geometric Reconstruction Methods Alternative formulations of free surface advection exist, where a "perfect" advection of a step profile is achieved with the use of geometric methods, albeit at a considerably higher computational cost. Recently, Ronby *et al* [22] introduced `isoAdvector` method, where accurate VOF advection is achieved through evaluation of VOF face flux with the use of a fluxing iso-surface for each face, Fig. 2. The method is explicit, practically limited by the Courant-Friedrichs-Levy (CFL) condition and significantly more expensive than conventional advection, but it produces remarkable results.

Geometric free surface advection scheme such as `isoAdvector` are used in cases where practical limitations in local mesh size exists and a clean interface is essential. A good example is a green sea simulation or slamming impact.

4.2.3. Level Set Methods As already noted, LS methods suffer from two drawbacks: lack of strong phase conservation and need for re-initialisation as a means to preserve unit gradient of the distance function. Vukčevic *et al* [12] introduce an alternative formulation of the LS, inspired by the properties of the terms in the PF equation which preserve self-similarity. Reformulated PF terms in the LS equation not only remove the need for separate re-initialisation but also improve the conservation properties compared with the original LS formulation written for the signed distance variable ψ:

$$\frac{\partial \psi}{\partial t} + \nabla\bullet(\mathbf{c}\psi) - \psi\nabla\bullet\mathbf{c} - b\nabla\bullet(\nabla\psi) = b\frac{\sqrt{2}}{\epsilon}\,\tanh\left(\frac{\psi}{\epsilon\sqrt{2}}\right), \tag{13}$$

where ψ is the signed distance function, $\mathbf{c}$ is the combined convective velocity:

$$\mathbf{c} = \mathbf{u} + b\frac{\sqrt{2}}{\epsilon}\tanh\left(\frac{\psi}{\epsilon\sqrt{2}}\right)\nabla\psi + b\kappa\,\frac{\nabla\psi}{|\nabla\psi|}, \tag{14}$$

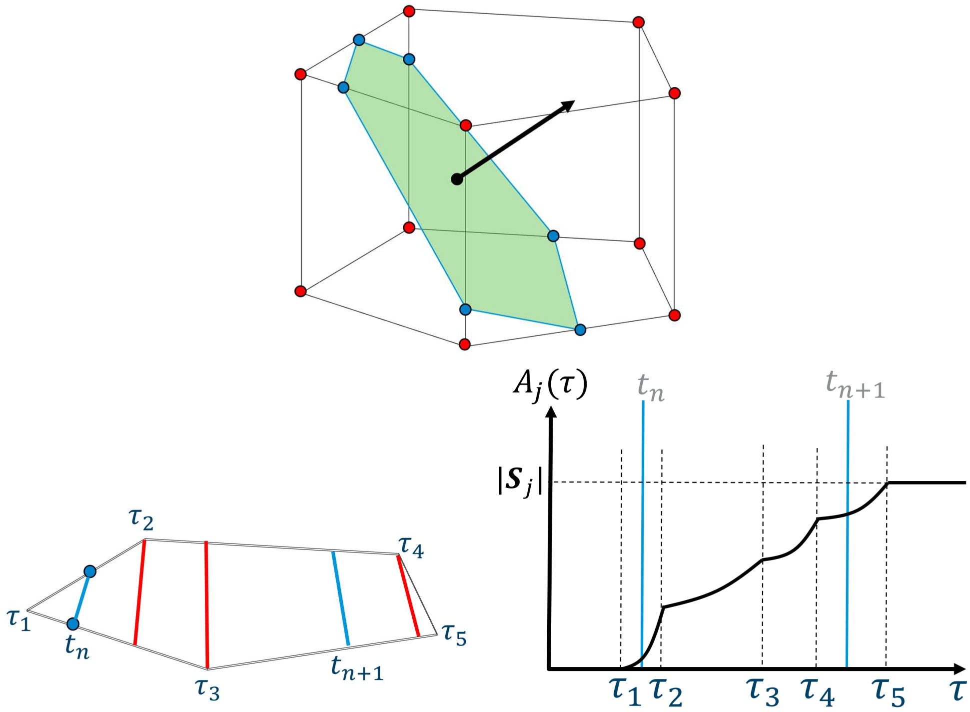

Figure 2. Calculation of the face flux using the `isoAdvector` scheme, [22].

where b is a numerical parameter used to smear out possible local singularities, ϵ is the smearing distance for the PF representation of the LS field and κ is the mean interface curvature. In the absence of curvature driven motion, as is the case in ship hydrodynamics free surface flows, the three terms on the r.h.s. of (13) numerically maintain the signed distance profile implicitly during the transport [18].

Although numerically more complex, this equation incorporates self-similar advection through a balance of additional convection and diffusion terms and is treated in a fully implicit manner. Furthermore, as ψ is not a bounded variable, it is suitable for use in solution decomposition strategies, as described below.

4.3. Hydro-Mechanical Coupling

The case of a floating body supported in free surface flow is often encountered in naval hydrodynamics. Here, the force balance on a floating body is handled via a 6-Degrees-of-Freedom (6-DOF) Ordinary Differential Equation (ODE) with body forces being balanced by pressure and friction acting on the wetted surface. In response to the 6-DOF solution, wall boundary is moved in the CFD domain, changing the hydrostatic and hydrodynamic response in a non-linear manner.

Hydro-mechanical coupling is resolved via outer flow – 6-DOF iterations at considerable cost. A representative number of outer iterations on the system is 6 to 8, with a total of 24 pressure solutions within a single time-step. Gatin *et al* [23, 24] propose an enhanced 6-DOF to pressure coupling, where the boundary motion is included into the fluid pressure solution. The new algorithm resolves the hydro-mechanical coupling in as little as 2 outer and 2 PISO iterations, reducing the CPU time by a factor of 3.

IOP Conf. Series: Materials Science and Engineering **276** (2017) 012009 doi:10.1088/1757-899X/276/1/012009

5. Wave Generation and Propagation

With numerical techniques described above it is possible to assemble an efficient CFD tool for free surface flows. In practical applications, a significant part of structural loading results from wave action: it is thus necessary to develop a wave initialisation technique within the CFD framework. The objective is to introduce a realistic wave elevation and velocity either for monochromatic waves or wave spectra into the CFD simulation with minimum distortion.

Wave reflection would naturally occur at the outlet and side boundaries may potentially destroy the solution; it is necessary to devise an appropriate practice to remove undesired wave reflections from domain boundaries.

5.1. Waves and Relaxation Zones

For simple monochromatic waves, waves can be introduced via a conventional wave boundary condition. For more complex waves, *e.g.* in wave focusing or irregular sea states, a simple boundary condition does not provide necessary fidelity. Also, removing reflection at outlet boundaries via a boundary condition works well only when the wave-length and speed of propagation is known. For those reasons, a *relaxation zone* approach is preferred [25], where the CFD solution is blended with the prescribed "far field" wave state within a relaxation zone.

In our simulations we use a modified form of relaxation, where the blending is performed implicitly, at the level of discretisation matrix [12, 20, 26]. The implicit method is not only more consistent and robust, but also works well with relaxation zone length of the order of one significant wave length. Detailed verification and validation data for the use of implicit relaxation zones is reported in [27, 20].

In sea-keeping and global performance simulations, we use relaxation zones with a prescribed "far-field" wave theory across all boundaries. This allows generality in a sense that the waves can propagate in arbitrary directions. With this approach, it is also easier for the analyst to establish how accurately the waves are transported through the CFD region, as any dissipation (loss of amplitude) or dispersion (phase error) becomes immediately visible in the results.

5.2. SWENSE Formulation

In sea-keeping or global performance CFD, far field wave condition carries a bulk of the flow solution. The effect of the floating body on the wave field, via diffracted and radiated waves presents only a fraction of the solution – if the non-linearity may be considered low. It is therefore reasonable to use the SWENSE solution decomposition technique [28, 12], where the CFD solution is decomposed without simplification into two parts:

$$\xi = \xi_I + \xi_P, \tag{15}$$

where

- ξ_I is the incident field, representing the far field wave condition and
- ξ_P is the perturbation field, representing the difference between the complete non-linear flow solution and the chosen incident field.

Under such decomposition, and with a wisely chosen incident field ($|\xi_P| << |\xi|$), it is possible to achieve superior wave propagation and still give a full non-linear CFD solution. In a CFD simulation, the velocity field can be decomposed in a straightforward manner. To describe the free surface position, Vukčević chooses to apply (15) to the LS equation to avoid boundedness problems with other colour functions (*e.g.*VOF, PF). Particular attention is given to the cases where the prescribed incident solution p_I, U_I, ψ_I does not strictly satisfy the discrete form of the Navier-Stokes equations, as it is necessary to compensate for the compatibility error [12].

6. Regular and Irregular Waves

The remaining task in setting up a successful CFD simulation of wave loads consists of prescribing the incoming sea state.

6.1. Regular Waves

In first attempts, analytical forms such as first-, second- and fifth-order Stokes waves were used, with optional superposition of current and/or sea bottom boundary layer profiles. Additional wave forms, such as solitary and c-noidal waves are also implemented.

It was noted that almost all regular waves can be accurately modelled with the fully nonlinear stream–function wave theory [29], at a negligible computational cost compared to CFD. Since the method takes into account the presence of a current implicitly, stream–function wave theory is a preferred choice.

6.2. Wave Focusing and Wave Spectra

While regular wave simulations are already of practical interest, industrial needs regularly require "design waves" or freak waves, modelled as a linear superposition of regular waves optimised to give a desired freak wave height. A freak wave assembled from a wave package is created from a given incoming wave spectrum, by tuning phase shifts over *e.g.* 100 wave components. The resulting wave load is then used in further structural analysis: it is therefore essential to correctly assemble such a wave package and ensure that its shape is preserved at the point of interest/impact.

To model realistic sea states which adhere to the desired spectrum, a selection of spectra is also implemented, This includes the Bretschneider, JONSWAP and Pierson-Moskowitz spectrum, with an additional option of providing an arbitrary list of wave components.

6.3. Higher-Order Spectra

During the propagation of wave spectra using CFD, a distortion of the spectra occurs due to non-linearities arising from wave-to-wave interaction and wave modulation. These non-linearities are present in nature as well and play a major role in evolution of the sea state. In order to resolve these non-linearities outside of the CFD simulation, Gatin *et al* gatinEtAl2016HOS implemented and validated a potential Higher-Order Spectrum (HOS) solver. The solver implements a pseudo-spectral method, solving for the non-linear free surface boundary conditions truncated at arbitrary (user–defined) non-linear mode. Compared to CFD, the HOS solver is extremely fast and is used to propagate a realistic (directional) sea state based on the given spectrum, which can then be screened for realistic freak wave events. The HOS freak wave is superior to wave focusing since it includes non-linear wave-to-wave interaction and single–wave modulation and can be accurately transported within the CFD domain. Furthermore, HOS can be used to calibrate the input spectrum in order to achieved the target spectrum in CFD. The method has been coupled to CFD, with extensive verification and validation [30].

7. Wave-Induced Loads

Having assembled the CFD solver capable of accurately propagating waves provided via the far-field implicit relaxation zones, a detailed study of prediction accuracy for wave-induced loads has been performed. The study starts with the analysis of wave loads on static structures and then proceeds with floating objects in regular and irregular seas.

7.1. Waves on Static Structures

Vukčević [20] provides a detailed verification and validation study for wave loads on a vertical surface-piercing cylinder for various wave steepness, providing guidance on mesh resolution in

IOP Conf. Series: Materials Science and Engineering **276** (2017) 012009 doi:10.1088/1757-899X/276/1/012009

the vertical and horizontal direction (with respect to the wave height and length), time-step size and number of wave encounters necessary to achieve periodicity. The solution is analysed in terms of wave propagation accuracy without the cylinder and in evaluation of higher-order forces resulting from wave diffraction from the cylinder.

7.2. Floating and Flexible Structures

Having established good force prediction for a static object, a comprehensive sea-keeping study for a ship in head and oblique waves was reported at the Tokyo 2015: A Workshop on CFD in Ship Hydrodynamics [31, 32]. The study starts with the evaluation of steady resistance, sinkage and trim for the KCS hull [33], followed by the verification and validation study for the same hull in head waves of various wave lengths and oblique waves in various directions. For all simulations, mesh resolution and periodic uncertainties are reported, while the time-step size and hydro-mechanical coupling uncertainties are presented for a representative case. As a guidance, a mesh of 1.6 million cells provides sea-keeping results with uncertainties of the order of 4%, with simulation time of approximately 15 minutes per wave encounter period on a small HPC cluster [34].

7.3. Mooring Systems, Propulsion and Sailing

In order to expand the applicability of the method, handling of a mooring systems and propulsion modelling is added to the package. Mooring forces are handled via the interface to the 6-DOF ODE solver, either in terms of linear/torsional spring/damper system, or a direct interface to external mooring model software. Model performance and technical readiness of such a model for practical offshore floater design has been reported by Kim *et al* in [35].

7.4. Green Sea Loads

As the next step, the issue of green water loading is considered. Here, two problems arise. First, the frequency of occurrence of water-on-deck events is relatively rare and a statistical/screening method is required to select specific event of sea-keeping in irregular seas. Secondly, a highly accurate method for the description of free surface on (necessarily) under-resolved meshes is needed. The focus here is in the surface capturing without numerical smearing errors, needed to accurately capture water impact on the structure.

Gatin [19] reports a comprehensive procedure for green sea load evaluation, combining the screening of the HOS spectral solution for irregular seas, sea-keeping simulation for ship motion for the selected events and highly accurate impact simulation.

In green water impact simulation, the `isoAdvector` geometric reconstruction scheme by [22] is used, with remarkable results.

Gatin [36] reports the comparison of numerical and experimental study for the green water impact on a fixed barge, based on experimental results by Lee *et al* [37] for 9 incident waves. A combination of advanced numerical practices produces remarkable good experimental agreement, shown as a comparison of pressure peaks and pressure force integrals for 10 measuring locations in Fig. 3.

Error bars on the CFD results in the figure show the combined periodic and mesh uncertainty, while the experimental error bars show periodic and measuring uncertainty.

8. Full Scale Effects

For a number of years, sea-keeping CFD has been performed at model scale. While this can be somewhat justified by the lack of full-scale experimental data, the need for full-scale data by naval architects can no longer be ignored. For CFD studies, dealing with full-scale ships and floating platforms brings further modelling problems, related with self-propulsion, course-keeping and full-scale turbulence effect.

Vukčević *et al* [38] report the results for a full scale benchmark simulation of a general cargo carrier used in the 2016 Workshop on ship scale hydrodynamic computer simulation. The CFD

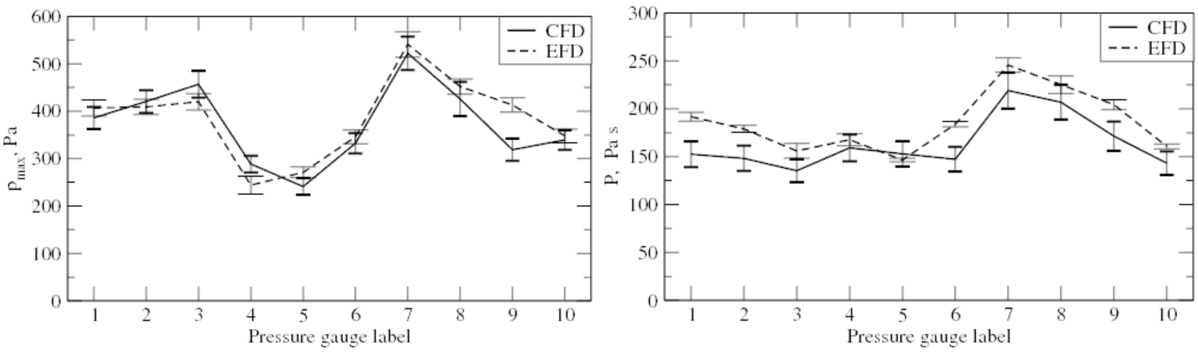

Figure 3. Simulation of Green Sea Impact: pressure peaks (left) and force integrals (right).

model included a full-scale ship hull propelled using the actuator disk propeller model under free sailing condition. The simulation results were compared with sea trial data. The presented results show that the self-propulsion point has been reached, predicting the ship speed to 0.3%, with grid uncertainty of 0.02 knots against the measured mile data, which is remarkable. A test was subsequently repeated on a different geometry in collaboration with Uljanik shipyard, with similar results.

9. Statistically Significant Modelling

At the current point of development, simulation of statistically significant results in a prescribed irregular sea state is coming into focus. Typical simulations will involve global performance CFD simulations of a moored floating platform or FPSO in a 3-hour storm, or a *safe return to port* simulation for a passenger ship.

Safety analysis and novel design criteria, such as designing an optimal ship hull for a given sea state (as opposed to optimisation for steady resistance in calm seas) will pose new challenges on robustness, accuracy and speed of simulation in naval hydrodynamics and off-shore CFD.

10. Summary

This paper presents the state-of-the-art of naval hydrodynamics CFD as seen by the author. While good progress has been made in topics of structural loading by regular waves and steady resistance of ship hulls, significant challenges remain. A contribution to the applicability and practical use of CFD in ship and off-shore design will be made by a combination of improved numerical practices, extensive validation and verification and some improvement in HPC hardware performance.

This paper reports on improvements in numerical discretisation, free surface capturing, coupling of simplified far-field wave models with CFD, generation and propagation of regular and irregular waves and detailed validation and verification of the CFD tools for loads on static structures, sea-keeping in regular waves, simulation of freak wave impact and green sea loads. Further challenges lie in the statistical modelling of irregular seas, where significant further work is needed.

References

[1] Ubbink O and Issa R 1999 *J. Comp. Physics* **153** 26–50
[2] Jasak H 1996 *Error analysis and estimation in the Finite Volume method with applications to fluid flows* Ph.D. thesis Imperial College, University of London
[3] Queutey P and Visonneau M 2007 *Comput. Fluids* **36** 1481–1510
[4] Vukčević V, Jasak H and Gatin I 2017 *Computers & Fluids* **153** 1 – 19 ISSN 0045-7930
[5] Muzaferija S and Perić M 1997 *Numerical Heat Transfer, Part B: Fundamentals* **32** 369–384
[6] Jasak H and Tuković v 2007 *Transactions of FAMENA* **30** 1–18
[7] Aulisa E, Manservisi S, Scardovelli R and Zaleski S 2003 *J. Comput. Phys* **192**
[8] Røenby J, Bredmose H and Jasak H 2016 *Open Science* **3**
[9] Sussman M and Fatemi E 1999 *SIAM J. Sci. Comput.* **20** 1165–1191
[10] Olsson E and Kreiss G 2005 *J. Comput. Phys.* **210** 225–246
[11] Olsson E, Kreiss G and Zahedi S 2007 *J. Comput. Phys.* **225** 785–807
[12] Vukčević V, Jasak H and Malenica v 2016 *Ocean Engineering* **121** 37–46
[13] Sun Y and Beckermann C 2007 *J. Comput. Phys.* **220** 626–653
[14] Sun Y and Beckermann C 2008 *Physica D* **237** 3089–3098
[15] Tuković v and Jasak H 2012 *Computers and Fluids* **55** 70–84
[16] Gómez P, Hernández J and López J 2005 *Int. J. Numer. Methods Eng.* **63** 1478–1512
[17] Hartmann D, Meinke M and Schröder W 2008 *J. Comput. Phys.* **227** 6821–6845
[18] Folch R, Casademunt J and Hernández-Machado A 1999 *Phys. Rev. E* **60** 1724
[19] Gatin I, Vukčević V and Jasak H 2017 *12th OpenFOAM Workshop*
[20] Vukčević V, Jasak H and Malenica S 2016 *Ocean Engineering* **121** 76–88
[21] Rusche H 2003 *Computational fluid dynamics of dispersed two-phase flows at high phase fractions* Ph.D. thesis Imperial College, University of London
[22] Roenby J, Bredmose H and Jasak H 2016 *Open Science* **3**
[23] Gatin I, Vukčević V, Jasak H and Rusche H 2017 *Ocean Engineering* **143** 295–304
[24] Gatin I, Vukčević V and Jasak H 2017 *VII International Conference on Computational Methods in Marine Engineering (MARINE 2017)* pp 663–669
[25] Jacobsen N, Fuhrman D and Fredsoe J 2012 *International Journal for Numerical Methods in Fluids* **70** 1073–1088
[26] Vukčević V, Jasak H and Malenica S 2015 *Proceedings of the 18th Numerical Towing Tank Symposium, NUTTS*
[27] Vukčević V 2016 *Numerical modelling of coupled potential and viscous flow for marine applications - in preparation* Ph.D. thesis Faculty of Mechanical Engineering and Naval Architecture, University of Zagreb PhD Thesis
[28] Ducrozet G, Engsig-Karup A P, Bingham H B and Ferrant P 2014 *J. Comput. Phys.* **257** 863–883
[29] Rienecker M M and Fenton J D 1981 *J. Fluid. Mech.* **104** 119–137
[30] Gatin I, Vukčević V and Jasak H 2016 *Ocean Eng. - under review*
[31] Vukčević V and Jasak H 2015 *Proceedings of the Tokyo 2015: A Workshop on CFD in Ship Hydrodynamics* vol 3 pp 437–442
[32] Vukčević V and Jasak H 2015 *Proceedings of the Tokyo 2015: A Workshop on CFD in Ship Hydrodynamics* vol 3
[33] Gatin I, Jasak H and Vukčević V 2015 *Proceedings of the Tokyo 2015: A Workshop on CFD in Ship Hydrodynamics* vol 3 pp 431–436
[34] Vukčević V, Jasak H, Gatin I and Malenica S 2016 *Proceedings of the 31st Symposium on Naval Hydrodynamics*
[35] Kim J W, Jang H, Bauet A, O'Sullivan J, Lee S, Kim B, Reed A and Jasak H 2016 *Offshore Technology Conference* (Houston, TX, USA) oTC-27294-MS
[36] Gatin I, Vukčević V, Jasak H, Seo J and Rhee S 2017 *Ocean Engineering* **In Press**
[37] Lee H H, Lim H J and Rhee S H 2012 *Ocean Engineering* **42** 47–60 ISSN 0029-8018
[38] Vukčević V, Jasak H, Gatin I and Uroić T 2017 *Proceedings of the VII MARINE 2017 conference* pp 1052–1061