THE NAVAL HYDRO PACK: CURRENT STATUS AND CHALLENGES

VUKO VUKČEVIĆ[1,2], INNO GATIN[1,2], HRVOJE JASAK[1,2]
[1]*University of Zagreb, Croatia,*{*vuko.vukcevic, inno.gatin, hrvoje.jasak*}*@fsb.hr*
[2]*Wikki Ltd, United Kingdom,* {*v.vukcevic, i.gatin, h.jasak*}*@wikki.co.uk*

Keywords: *Marine and Offshore Hydrodynamics, Interface Capturing Schemes, Ghost Fluid Method, Wave Modelling, Hydro–Structural Coupling, Dynamic Mesh*

This paper serves as an overview of the numerical modelling technology within the Naval Hydro Pack, a software package specifically designed for efficient simulations of various marine hydrodynamic problems. The Naval Hydro Pack is based on `foam-extend`, a community driven fork of the OpenFOAM software. The goal of this paper is to provide an overview of numerical models required for free surface flows in marine hydrodynamics, where we intend to openly discuss drawbacks and advantages of each of our methods.

Introduction

We start by considering the final goal of the Naval Hydro Pack: we would like to be able to reliably calculate the most complicated problem in marine hydrodynamics with reasonable computational resources and sufficient accuracy. The most difficult problem we can currently think of is a self–propelled ship performing some kind of manoeuvre in a severe storm. To add a an additional layer of complexity, imagine that the ship is an ultra large container carrier where the structural response and hydrodynamic excitation are interdependent. Let us now take a "divide–and–conquer" approach of this complicated problem and identify the underlying challenges:

1. The first obvious challenge is the modelling of the two–phase flow (water and air), where we need to accurately take into account free surface kinematics and dynamics,
2. The second challenge is the efficient modelling of gravity waves, propagation and prevention of wave reflection,
3. The third challenge is associated with the hydro–structural coupling where the ship naturally responds to hydrodynamic forces acting on it. Even if we consider the ship as a rigid body, the hydro–structural coupling is highly nonlinear and therefore efficient strategies for resolving this coupling need to be considered.

In the following text, we discuss our solutions to these challenges without going into details, while providing references to all topics.

Free Surface Modelling

In marine hydrodynamic flows, the free surface is dividing two immiscible phases: water and air. Two distinct problems arise in accurate numerical handling of the free surface: i) How to represent and transport the free surface and ii) Once the free surface location is known, how to obey the jump conditions due to different physical properties of air and water?

Interface capturing schemes

In the Naval Hydro Pack, three different interface capturing schemes can be used to advect the free surface:

1. Algebraic Volume–of–Fluid Method (A–VOF),
2. Implicitly Redistanced Level Set Method (IR–LS) [1],
3. Geometric Volume–of–Fluid Method (G–VOF) [2].

The advantages and drawbacks of these schemes are summarized in Table 1, where the reader can easily deduce that there is no "perfect" tool and some trade–off is always necessary. Here, we briefly discuss our best practice guidelines for different types of marine hydrodynamic problems. The A–VOF scheme is suitable for steady resistance and seakeeping simulations where large time–steps are required and the perfect mass conservation is desirable. In cases where unphysical numerical smearing of the interface is present with the A-VOF scheme due to complex flow field or really low–quality mesh, we tend to use the IR–LS method [1]. The method proved to be excellent in preventing numerical ventilation when considering high speed craft. Although the method is not strictly mass conservative, this does not seem to affect the results we are interested in: forces and motions. The G–VOF scheme [2] is by far the most accurate scheme and therefore we

Table 1: Comparison of different interface capturing schemes as implemented in the Naval Hydro Pack.

Advection scheme	**A-VOF**	**IR-LS**	**G-VOF**
Mass conservation	Machine tolerance	Discretisation error	Machine tolerance
Courant number limit	No	No	Yes, $C_o < 1$
Control of interface smearing	Poor	Excellent	Excellent

often use it for violent free surface flows where we seek accurate interface resolution, *e.g.* green water simulations [3]. The obvious drawback is that it requires a Courant number lower than 1.

Ghost Fluid Method for interface jump conditions

Once the free surface location is known, the discontinuity in density and consequently dynamic pressure needs to be taken into account in an accurate way. If one uses standard discretisation practices where the dynamic pressure and density gradient are coupled within the momentum equation, this will cause spurious velocities near the interface in the lighter phase [4] (even without surface tension). The phenomenon can also cause difficulties with advection of the interface because the velocity field is non–physical near the interface. The kinematic and dynamic boundary conditions between water and air can be taken into account with interface–corrected discretisation schemes with the Ghost Fluid Method (GFM) [4]. A similar approach is used by Queutey and Visonneau [5] in the ISIS–CFD software (the CFD solver behind NUMECA's FINE/Marine suite). Currently, our implementation handles dynamic pressure and density jumps across the interface, while the tangential stress balance is still approximated by calculating the velocity gradient using standard Finite Volume discretisation. All interface capturing schemes are used alongside the GFM method in the Naval Hydro Pack. For backward compatibility reasons, the old formulation without the GFM is still available, although we rarely use it.

There is one difficulty with the GFM that we have recently experienced. Since the GFM assumes infinitesimally discontinuous distribution of the density and dynamic pressure, the simulation becomes unstable if water entrains air in a way that the air should be compressed. This is actually the drawback of the incompressibility assumption and not the GFM method. Currently, we are working on extending the GFM for two phase flows where one phase is fully incompressible (water) and the other is compressible with isentropic equation of state (air).

It is equally important to note the indirect benefits of the GFM. As the velocity field across the free surface is continuous and there are no spurious air velocities in air, we observe two advantages: i) The advection of the free surface with all methods is much more stable since the flux is well defined across the interface and ii) The maximum Courant number is significantly lower, enabling faster simulations.

Wave Modelling

In order to efficiently model gravity water waves in the CFD domain, different strategies can be used. In the Naval Hydro Pack, we extend the relaxation zone approach by Jacobsen *et al.* [6] in order to have implicit blending during the transport [1]. The relaxation zones provide an efficient and straightforward tool that we can build upon in future to couple our CFD model with CPU time efficient potential flow models. The approach is also favourable for considering unbounded domains (*e.g.* ships sailing in ocean) and directional sea states. Since the potential flow solution has negligible cost compared to CFD, we prefer to use fully nonlinear stream function wave theory for monochromatic waves and Higher Order Spectrum (HOS) method [7] for irregular sea states. The HOS is quite favourable since it accounts for nonlinear wave modulation and wave–wave interaction, so there is no need to resolve these within CFD.

In the Spectral Wave Explicit Navier–Stokes Equations (SWENSE) method as implemented in the Naval Hydro Pack [1], the waves are introduced in the whole CFD domain. This approach gives us the possibility of calculating only the nonlinear perturbation around the explicit incident wave, instead of calculating the total fields using the standard approach and introducing waves only within the relaxation zones. Since the incident wave field is explicit in the SWENSE method, it slightly facilitates the wave propagation in CFD domain. However, for violent free surface effects such as green water, it is reasonable to expect that the flow solution will be significantly different from the incident wave, making the SWENSE method unsuitable for this type of problems.

Hydro–Structural Coupling

The previous two sections give us a variety of tools to handle the two–phase flow, while the missing bit is the coupling of the flow field with the motion of a ship or another offshore object. Two distinct challenges can be identified when considering the hydro–structural coupling: i) Coupling strategy between the fluid–flow and 6 DOF equations of motion and ii) Efficient handling of dynamic mesh.

Algorithms for hydro–structural coupling

In Finite Volume CFD, almost all hydro–structural coupling algorithms are partitioned, meaning that the flow field and 6 DOF equations of motion are solved one after another in an iterative manner. In the Naval Hydro Pack, the 6 DOF equations are strongly coupled to the fluid flow within the nonlinear (PIMPLE) loop, where a sufficient number (usually six or more for seakeeping) of nonlinear corrector steps is necessary to converge the solution within a time–step. We have implemented an enhanced strategy where the 6 DOF equations are additionally integrated after each pressure correction step. The approach provides significantly improved convergence of 6 DOF and flow field at a negligible CPU time cost. The benefit of this approach is demonstrated in Gatin *et al.* [8], where almost the same motion amplitudes are obtained by using 2, 4, 6 or 8 nonlinear correctors for the seakeeping of the KCS model. This allowed us to speed-up our seakeeping simulations up to three times without sacrificing the accuracy.

Recently, we have also developed a monolithic coupling approach, where the 6 DOF equations are solved as a constraint within the pressure equation. Similar to the enhanced coupling, the monolithic coupling improves the convergence with lower number of nonlinear correctors, as presented on Figure 1 where the heave motion for seakeeping of the KCS model is depicted. The monolithic approach proved to be quite favourable for motions with high acceleration (*e.g.* seakeeping of high speed planning hulls) and high added mass (relative to the mass of the ship). However, it should be stated that for the seakeeping of displacement ships, this approach is rather excessive: although it can be used confidently, the benefit will be minor compared to the enhanced approach [8]. The numerical details and benefits of this approach are described by Jasak *et al.* [9].

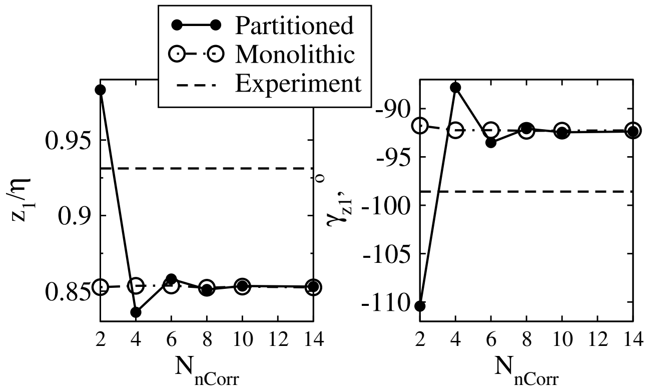

Figure 1: Heave amplitude and phase for different number of nonlinear correctors, KCS model, C5 case, Tokyo Workshop [10]

Dynamic Mesh Handling

Once the 6 DOF equations are solved and the new position of the ship is obtained, one needs to efficiently handle the dynamic mesh motion. Different approaches exist in the Naval Hydro Pack and `foam-extend`, which are summarized in Table 2. The first option is the simplest: the domain is moved as a rigid body. The method is simple to implement and robust, while it does not allow multiple bodies and is not suitable for fairly large motions. The second method is mesh deformation, where we prefer to use an algebraic approach where the mesh is moved rigidly in the vicinity of the body and the deformation slowly decays towards farfield boundaries. The approach is as efficient as the domain motion. Although it can also have difficulties with large amplitude motions, the method is suitable for most of the standard marine hydrodynamic problems. Both domain motion and mesh deformation strategies cannot handle appendages that can move relative to the moving ship. In `foam-extend-4.1`, a significant improvements to the Immersed Boundary library and Overset Mesh library have been added, making these choices more suitable for marine hydrodynamic flows. Both methods offer extreme versatility in a sense that they can handle complex relative motions. The Overset Mesh method is more suitable for flows where viscous effects are important, while the Immersed Boundary method is favourable where the pressure effects are dominant and the viscous effects are of secondary interest. In current state, both methods are efficiently parallelised and work well, although the pre–processing stage is more demanding for the Overset Mesh since it requires careful consideration of the overlapping region. Both methods are in their intermediate stages of development in `foam-extend`, which practically means that the tools are working, although a significant portion of thorough verification and validation is missing, along with some additional minor development.

Conclusion and Future Work

A single conclusion can be drawn from this discussion and our experience during the past decade: there is no single best tool which will be able to efficiently tackle all the marine hydrodynamic problems. In our opinion, it is up to

Table 2: Comparison of different dynamic mesh strategies as implemented in the Naval Hydro Pack and `foam-extend`.

	Domain Motion	**Mesh Deformation**	**Overset Mesh**	**Immersed Boundary**
Complexity	Low	Low	High	High
Development stage	Mature	Mature	Intermediate	Intermediate
Robustness	High	High	Intermediate	Intermediate
Versatility	Low	Low	High	High

the experienced researcher and engineer to determine the most suitable combination of tools to tackle a given problem. Therefore, we have focused on developing and testing various interface capturing, interface handling, hydro–mechanical coupling and dynamic mesh methods in the Naval Hydro Pack. Where one method proves to be unsuitable, it is often more natural and less time consuming to switch to another method, rather than to ”improve” the original one.

Currently, our short term research and development covers the following topics: i) Improved turbulence modelling practices for marine hydrodynamics free surface flow. Ordinary turbulence models are derived and tuned for single phase flows, usually causing non–physical turbulent eddy–viscosity near the free surface [11], which needs to be reconsidered, ii) Extension of the GFM for tangential stress balance in order to more accurately calculate the velocity gradient near the free surface and iii) Extension of the GFM for incompressible/compressible two–phase flow.

In the long term, we will focus on more advanced dynamic mesh handling strategies: i) Improving the robustness of our automatic overlap assembly strategy in Overset Mesh library and ii) Further testing, verification and validation of the Immersed Boundary library for marine hydrodynamics. In order to tackle the problem discussed at the beginning of the paper, we are looking into obtaining the funding to develop a fully open–source framework for hydro–elastic computations where we would couple the Naval Hydro Pack with an open source FEM structural method.

Acknowledgments

The funding from Hyundai Heavy Industries under administration of Dr. Geon–Hong Kim and Bureau Veritas under administration of Dr. Quentin Derbanne is greatly appreciated. It allowed us to focus on developments that are industrially relevant and useful. We are also very thankful to our loyal partners that base their research and development on the Naval Hydro Pack: their demands, insightful comments and testing drive our development further in the right direction.

References

[1] V. Vukčević, H. Jasak, and S. Malenica, “Decomposition model for naval hydrodynamic applications, Part I: Computational method,” *Ocean Eng.*, vol. 121, pp. 37–46, 2016.

[2] J. Røenby, H. Bredmose, and H. Jasak, “A computational method for sharp interface advection,” *Open Science*, vol. 3, no. 11, 2016.

[3] Gatin, I. and Vukčević, V. and Jasak, H. and Seo, J. and Rhee, S.-H., “CFD Verification and Validation of Green Sea Loads,” *Ocean Engineering*, vol. 148, pp. 500–515, 2017.

[4] V. Vukčević, H. Jasak, and I. Gatin, “Implementation of the Ghost Fluid Method for Free Surface Flows in Polyhedral Finite Volume Framework,” *Comput. Fluids*, vol. 153, pp. 1–19, 2017.

[5] P. Queutey and M. Visonneau, “An interface capturing method for free–surface hydrodynamic flows,” *Comput. Fluids*, vol. 36, pp. 1481–1510, 2007.

[6] N. G. Jacobsen, D. R. Fuhrman, and J. Fredsøe, “A wave generation toolbox for the open-source CFD library: OpenFoam®,” *Int. J. Numer. Meth. Fluids*, vol. 70, no. 9, pp. 1073–1088, 2012.

[7] I. Gatin, V. Vukčević, and H. Jasak, “A Framework for Efficient Irregular Wave Simulations Using Higher Order Spectral Method Coupled With Viscous Two Phase Model,” *Journal of Ocean Engineering and Science*, vol. 2, pp. 253–267, 2017.

[8] I. Gatin, V. Vukčević, H. Jasak, and H. Rusche, “Enhanced coupling of solid body motion and fluid flow in finite volume framework,” *Ocean Eng.*, vol. 143, pp. 295–304, 2017.

[9] Jasak, H. and Gatin, I. and Vukčević, V., “Monolithic Coupling of the Pressure and Rigid Body Motion Equations in Computational Marine Hydrodynamics,” *J. Marine Sci. Appl.*, pp. 1–7, 2017.

[10] “Tokyo 2015: A Workshop on CFD in Ship Hydrodynamics,” http://www.t2015.nmri.go.jp/, 2015, [Online; accessed 20 August 2015].

[11] B. Devolder, P. Rauwoens, and P. Troch, “Application of a buoyancy-modified $k - \omega\ SST$ turbulence model to simulate wave-run up around a monopile subjected to regular waves using OpenFOAM,” *Coastal Eng.*, vol. 125, pp. 81–94, 2017.