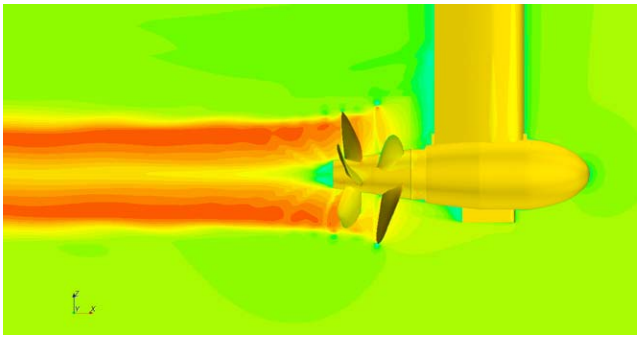
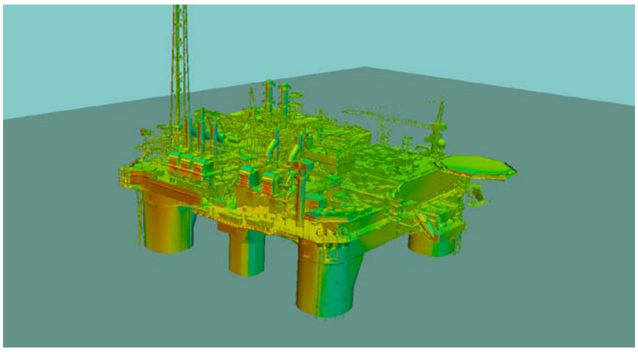
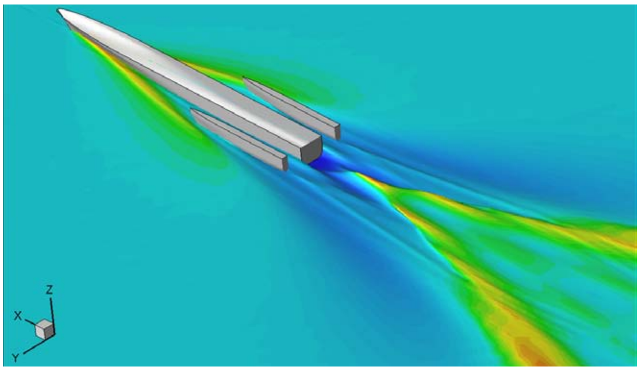
MARINTEK
Norwegian Marine Technology Research Institute
No. 2 • May • 2012
REVIEW

Computational Fluid Dynamics (CFD) as a performance evaluation tool at MARINTEK

A rational combination of physical and numerical modeling approaches has always been MARINTEK's strategy in its research and commercial services. Computational Fluid Dynamics (CFD) is a powerful tool that is increasingly being used in engineering simulations of fluid flows: prediction of propulsor performance, wind and current loads and wake analyses are just a few applications of CFD in marine hydrodynamics.







MARINTEK has been at the cutting edge of the development and use of numerical tool to model a wide range of fluid flows for several decades. Both viscous and potential methods, commercial, open-source and in-house codes are used in our projects, and are thoroughly validated for each application by comparisons with model test results.

CFD is generally used as a design-support tool and offers a complementary strategy to model testing. In fact, CFD is particularly suitable for providing a detailed 3D description of pressure fields and flow conditions, providing important insights into physical phenomena. In turn, advanced model testing and full-scale measurements each represent valuable ways to validate computational results, to verify numerical approaches and techniques, and to test different solvers and/or numerical schemes.

MARINTEK employs its long experience and extensive know-how to provide the best combination of potential flow, CFD and experimental analyses customized for different stages of each project.

CONTENTS

ISSN 0801-1818

SINTEF

Optimisation of hull lines

›› Research Scientist Eloïse Croonenborghs
›› Research Scientist Lucia Sileo
›› Research Scientist Anders Östman

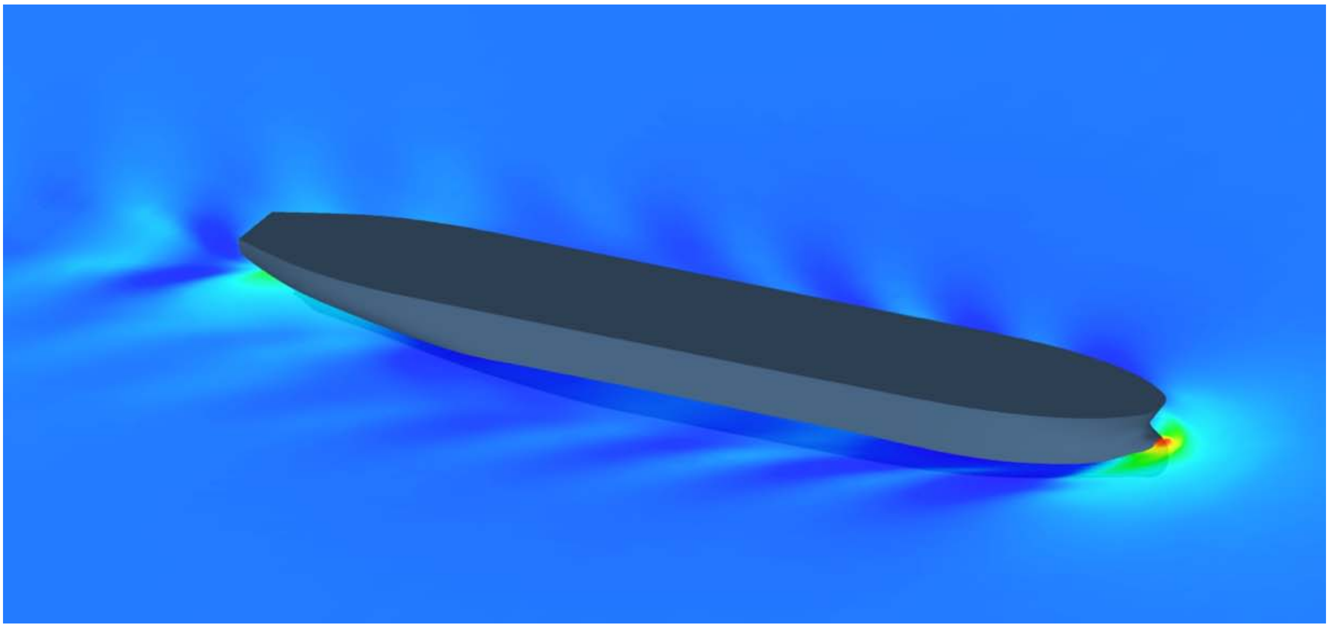


Figure 1: Wave pattern generated by a hull at constant speed in calm water.

Predictions of wave pattern and vessel performance in calm water using CFD computations are routinely made by MARINTEK. The improved insight into velocity and pressure distributions obtained by such computations offers all the information needed to optimise ship design through an effective process of hull-form refinement.

The shipbuilding industry and market are interested in designing hull forms that run faster, generate less noise and are more energy-efficient. A critical factor in predicting the performance of new hull designs is the ability to accurately simulate wave patterns around the hull form, as the waves have a major impact on the forces acting on the hull. In order to minimize those forces, hull shapes need to be studied and optimized in detail.

The small margins for improvement that are usually targeted in hull design nowadays require precision tools capable of discerning small differences between alternative designs. Custom modelling tools based on simplified numerical methods and assumptions cannot provide the required accuracy. For this reason, viscous computational methods are playing an increasingly important role in ship design, as they are capable of simulating turbulent flow around hulls, featuring thick boundary layers under the influence of strong cross-flow, pressure gradients, streamline curvatures, and stream-wise vortices. For this purpose, MARINTEK uses two different commercial RANSe (Reynolds Averaged Navier-Stokes equations) solvers, STAR-CCM+ and Fine/MARINE, to solve the equations of continuity and momentum of viscous flow

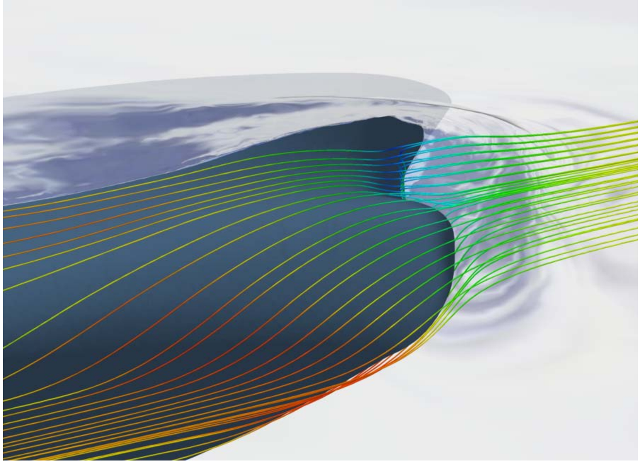


Figure 2: Streamlines representing the local flow field around a bow.

around the hull. Both codes include state-of-the-art turbulence models suitable for high Reynolds number flow fields, and they have been thoroughly validated for hull resistance prediction and free-surface modelling. They use VOF (Volume of Fluid) formulations which are capable of accurately capturing the sharp water-air interface.

CFD offers many inherent advantages that make it particularly suitable for an effective hull-lines optimisation process. It can handle arbitrary geometries, viscous flow and compressible flow. Detailed 3D descriptions of the flow conditions provide the information needed to minimize hull resistance in calm water at various speeds and under different operating conditions, which are factors of vital importance for the competitiveness of new designs. Changes in the design drawings can therefore be easily evaluated in terms of their hydrodynamic performance.

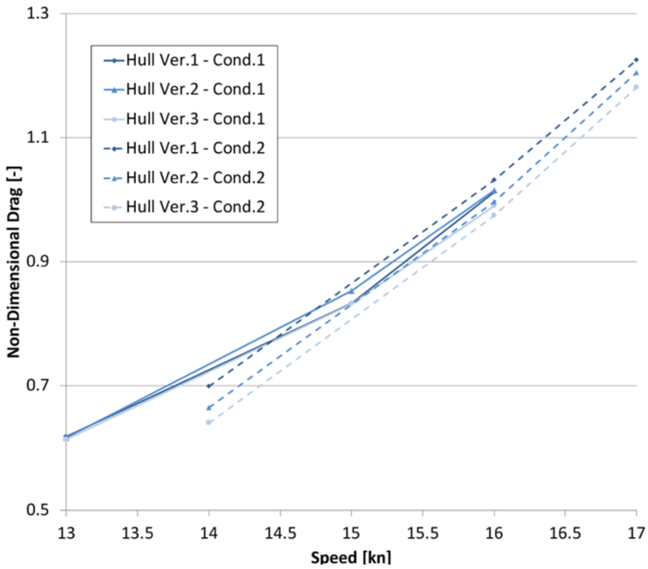


Figure 3: Optimization of the bulb and skeg design of a ship in order to minimize the propulsive power required at operating speeds. Three hull versions under two different loading conditions were studied.

A major advantage of using CFD in the design optimisation process is the detailed level of information of the flow field. Based on the results of a bare hull CFD study valuable insight in the optimum position of appendages such as brackets, bilge keels and headbox, can be given. By using CFD, problematic details of the hull design are also easily identified, which can include flow phenomena such as separated flow areas, local pressure variations, cross stream over sharp edges and areas with free surface spray or breaking waves.

Listed below are some examples of results that MARINTEK generally provides, based on the analysis of the computed flow field.

- Hull resistance
- Dynamic position, motion and acceleration
- Pressure distribution on the hull
- Free-surface wave pattern
- Limiting streamlines on the hull
- Local flow field in vicinity to appendages
- Wake field in the propeller plane.

Given on its long experience of hull design, MARINTEK can provide recommendations for the improvement of the design. By an iteration process alternating between CFD and revised drawings, the performances of several designs can be compared, thus gradually bringing the client toward the optimal solution. Because CFD provides answers more rapidly than building models and performing experiments, this process enables the designer to consider a much greater number of possible solutions than could ever be made in the past.

While calculations are especially performed for optimization of hulls and/or appendages during the early stages of the design process, model trials are usually carried out in order to evaluate the final design. This approach provides an opportunity to compare computed values of mean forces, as well as dynamic position, with experimental data, for the final design. The free-surface elevation at the bow and stern can be compared qualitatively with photographs taken during model testing in the towing tank.

This unique strategy whereby CFD is fully integrated into the design process can be applied to a very wide range of vessel types.

The figures shown here have been taken from the hull lines optimization process of a 72.900 DWT Shallow Draught Product Carrier designed by the Brazilian design company PROJEMAR; the CFD computations were performed last summer. Since 2004, PROJEMAR has utilised MARINTEK's expertise in hull lines optimization based on CFD analysis for several ship designs, including tankers, ore carriers and container ships, with excellent results.

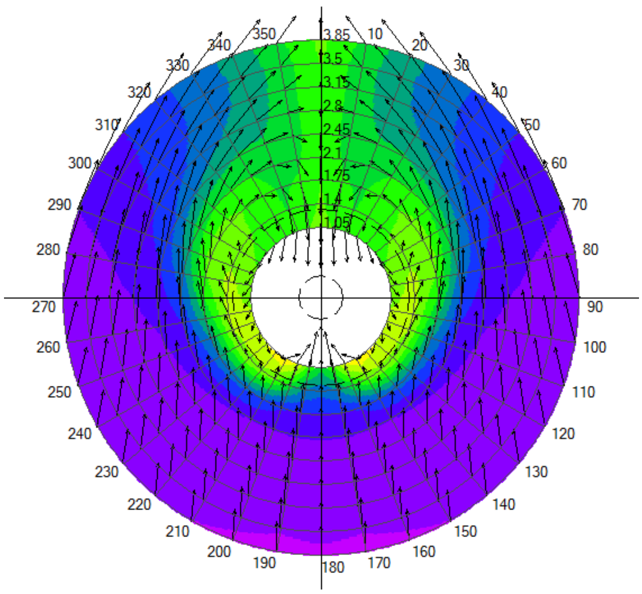


Figure 4: Wake field in the propeller plane of a Panamax vessel.

Hull resistance in oblique flow

›› Research Scientist Eloïse Croonenborghs

In ships which operate at an angle relative to the main direction of flow, such as tug-boats and sailing vessels, the hull appendages develop a lift force which is not aligned with the main flow. Besides an additional resistance component, the side force developed in oblique flow generates additional yaw, roll and pitch moments on the hull.

An accurate estimation of the yaw moment is therefore of great importance in order to ensure that the propeller unit can propel the ship in such special sailing configurations.

CFD computations enable the added resistance due to a steady drift angle, the resulting yaw moment on the hull, and the dynamic position of the ship in roll, pitch and heave to be quantified.

The figures shown here are taken from full-scale computations of a tug in oblique flow on behalf of the marine services company Buksér og Berging AS. In these simulations, the vessel is free to roll, pitch and heave.

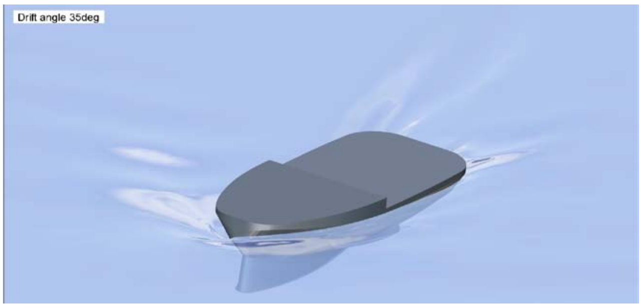


Figure 1: Wave pattern generated by a tug-boat advancing at a drift angle of 35 degrees.

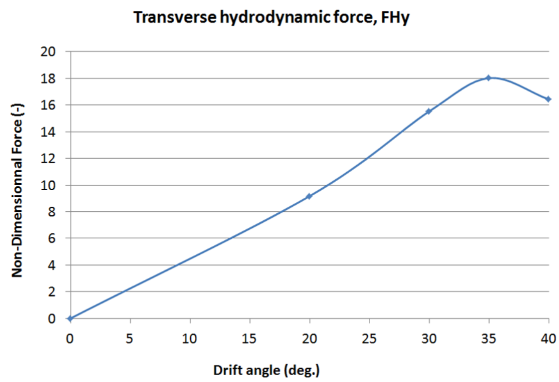


Figure 2: Transverse hydrodynamic force developed by a tug advancing at different drift angles.

Resistance for High-Speed Craft

›› Research Scientist Lucia Sileo

A detailed description of the flow field is even more important when considering high-speed vessels, i.e. vessels operating in the range of Froude numbers higher than about 0.4. In this case the buoyancy force is not dominant, while the hydrodynamic pressure becomes relevant, as it is approximately proportional to the square of the speed and depends on the flow around the hull.

At high speeds, the risk of cavitation on hydrofoils and propulsion systems also needs to be considered. CFD techniques are then often required in combination with model testing. Figures 1 and 2 show how RANS methods can be used to determine the power requirements of a waterjet-driven trimaran, as an example of the synergy between model tests and numerical methods.

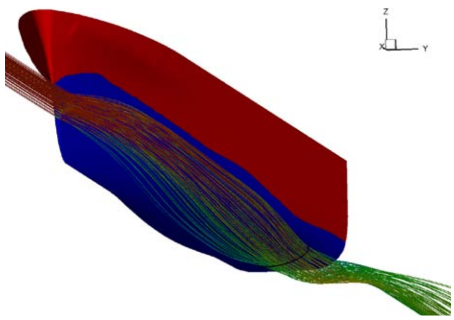


Figure 1: Determination of ingested momentum flux through the waterjet capture area on the main hull of a trimaran.

Figure 2: Trimaran, 35 knots. Wave pattern. Comparison with model tests.

Sea-keeping and added resistance in waves

›› Research Scientist Eloïse Croonenborghs

Figure 1: Wave profile along a ship hull

When vessel performance is being evaluated, behaviour in a seaway is as important as calm-water resistance. To evaluate ship responses and added resistance, MARINTEK performs unsteady CFD computations of ships advancing on a straight course in regular waves.

Hull resistance is traditionally estimated under calm-water conditions. This gives valuable information about the ship's performance at the early design stage. However, calm water is the exception at sea. Maintaining a desired heading in a seaway may induce large ship motions, increase resistance and reduce propulsive efficiency. Vessel responses and motions are very important from a safety point of view, while added resistance and loss of speed are economically important. The study of the behaviour and performance of vessels in waves is therefore an essential aspect of ship design.

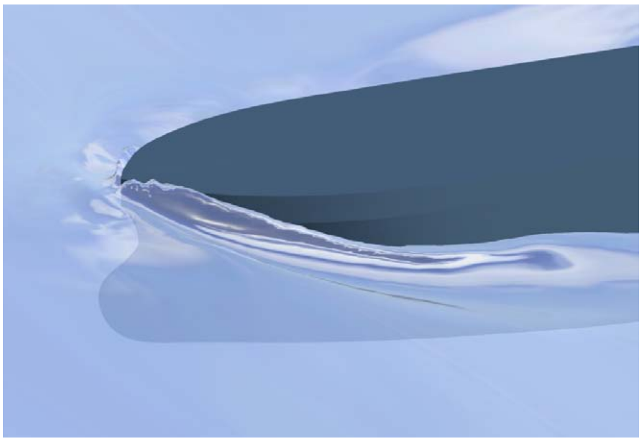


Figure 2: Wave breaking at the bow.

Since estimates of ship responses and added resistance are critical, it is essential to employ accurate prediction methods in the design process. Traditionally, the prediction of added resistance was done by analytical methods based on potential flow theory. Since these methods are useful for making gross estimates of added resistance, they have been widely used as practical design tools. However, a clear disadvantage of these methods is that they cannot account for nonlinear flow features such as waves breaking in the near-field of a vessel. Because these nonlinear features contribute significantly to the added resistance, analytical methods are often not accurate enough for the quantitative prediction of added resistance. CFD simulation methods offer the advantage of being able to deal directly with nonlinear flow phenomena without explicit approximations. This makes them capable of tackling problems with strong nonlinearity, such as the prediction of added resistance.

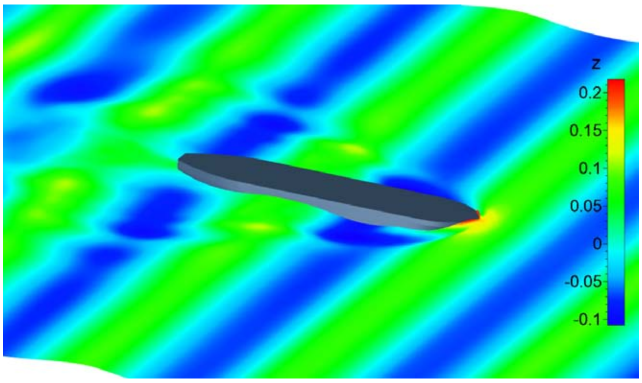


Figure 3: Wave pattern generated by a ship in head waves

At MARINTEK, unsteady CFD computations can be performed for a vessel advancing on a straight course in regular waves in order to evaluate its performance, including ship response and added resistance. Both free and fixed heave and pitch motions can be simulated for a wide range of important wavelengths. Such simulations require larger computing resources than calm-water resistance simulations because the grid must be very fine throughout the region occupied by the wave train in order to ensure good propagation of incident waves with no noticeable damping. However, very acceptable calculation times can now be obtained by running parallel computations on our cluster. Typical outputs from such simulations include time series of the hull resistance and ship motions. Movies can also be recorded over the duration of one period.

CFD modelling of marine propulsors

›› Senior Research Scientist Vladimir Krasilnikov
›› Research Scientist Lucia Sileo

The complexity of propeller geometry and the presence of rotational flows, with 3D boundary layers, strongly anisotropic structure of turbulence, high pressure gradients and fluid acceleration make the analysis of marine propellers one of the most challenging topics in marine hydrodynamics. Many numerical methods, with various levels of sophistication, are available and we describe their applications to the simulation of propulsor flows in this and the following sections.

Potential methods: AKPA

Work on the new generation of numerical tools for propulsors started about ten years ago and resulted in the development of the program AKPA. Based on potential flow theory, AKPA is widely used in marine propeller design applications by MARINTEK, NTNU, DNV and Norwegian manufacturing companies. The core of the system is the panel method analysis code that allows several different types of propulsors (open, ducted and podded propellers, propeller-rudder systems) to be simulated under steady and unsteady flow conditions, both with and without cavitation effects.

In the course of the years, the propulsor analysis algorithm has been extended through semi-empirical models to account for the effects of viscosity, which were derived on the basis of correlations with systematic viscous flow computations and measurements. More recently, the program has incorporated the two Euler equation solvers – axisymmetric and fully 3D – that allow for the estimation of effective inflow on propeller at specified design point.

Viscous methods

Besides the development of and improvements in the inviscid flow methods described above, recent years have seen a shift in emphasis towards the development and application of viscous flow methods. The use of Reynolds Averaged Navier-Stokes (RANS) equations is a more appropriate approach to the solution of real viscous flows due to their ability to model, while in averaged sense, the effects of turbulence by means of more or less sophisticated turbulence models.

Systematic validation, verification and calibration studies are routine practices in CFD. By combining experimental studies and numerical simulations, MARINTEK offers quality and accuracy in CFD simulations with the aim of achieving a better understanding of complex flow features and physical phenomena, including the mechanisms of cavitation.

CFD methods demonstrate very high accuracy in the prediction of propeller characteristics under a wide range of operating conditions, and they are capable of capturing and resolving different types of propeller and rudder cavitation with high degree of fidelity. Detailed analysis of interaction

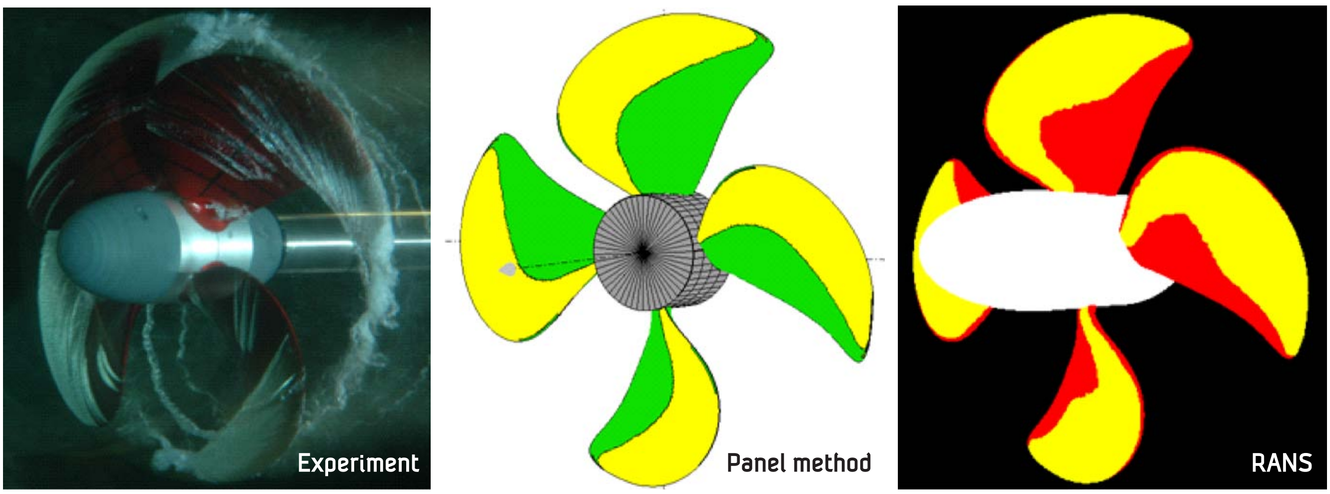


Figure 1: Experimental and numerical predictions of propeller cavitation under off-design conditions.

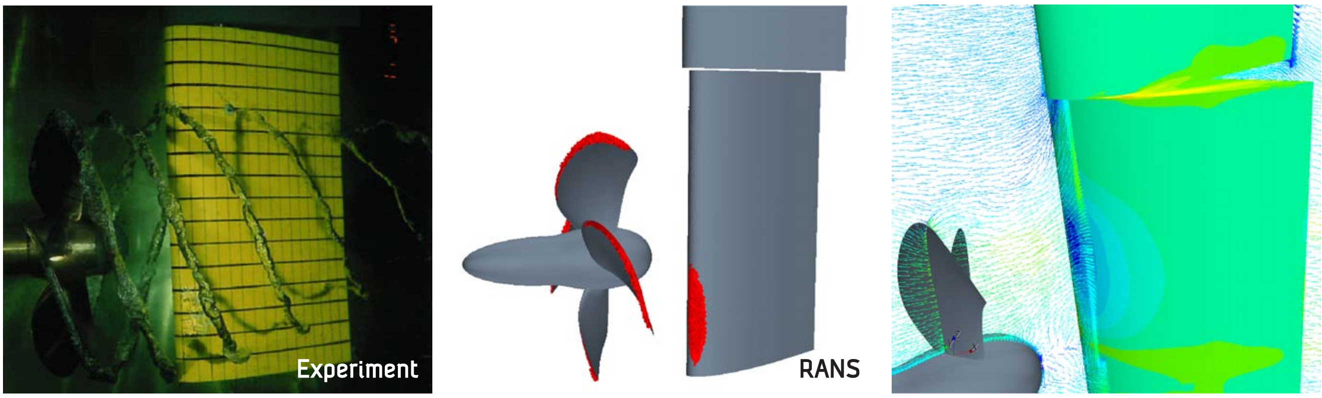
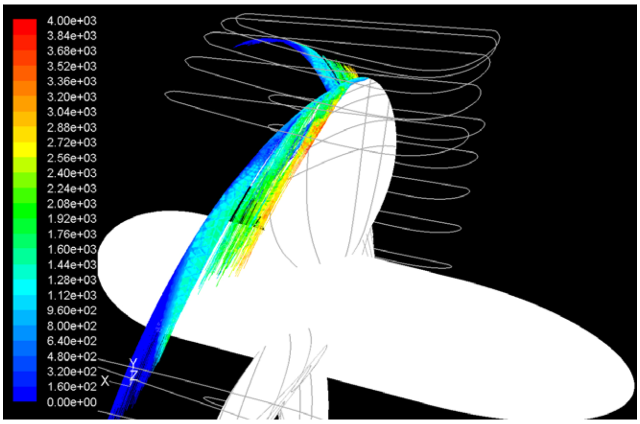


Figure 2: Experimental and numerical analyses of cavitation on a propeller with rudder under heavy loading.

and understanding of the underlying mechanisms of cavitation result in practical methods for the reduction of its most undesirable forms (Figures 1-2).

Numerical flow analyses can offer insight into flow phenomena that are not easily detected by even the most advanced PIV and LDV experimental techniques. For example, the use of a RANS solution enables us to study the genesis of gap flow and propeller tip vortex inside a duct by resolving the complex interactions that take place between the propeller tip vortex and duct boundary layer (Figure 3).

Flows around podded propellers operating at large heading angles are characterized by a high degree of separation and extended swirl domains that induce highly non-homogeneous inflow on the propeller, whereas propeller and pod experience strong interaction. Viscous flow methods not only reflect a qualitatively correct picture of the flow, but also predict forces on the unit with much higher accuracy than potential flow methods. For example, the RANS method has been shown to reproduce the asymmetry of the manoeuvring forces that act on a pushing podded thruster at positive and negative heading angles (Figures 4 and 5).



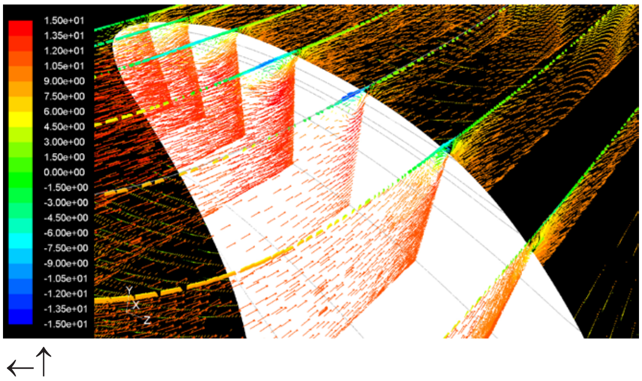


Figure 3: Formation of a double tip vortex on a ducted propeller.

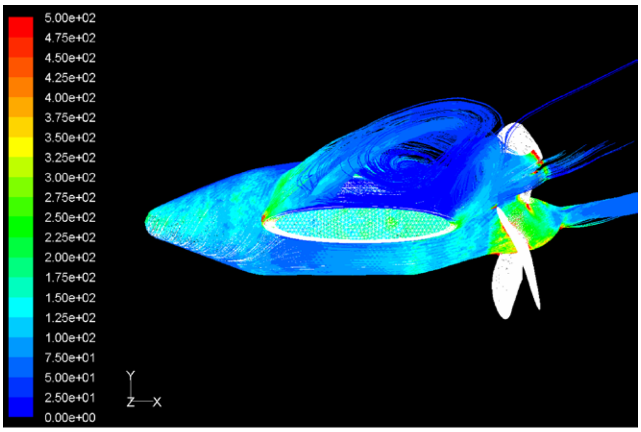


Figure 4: Flow pattern around a podded propeller at large heading angle.

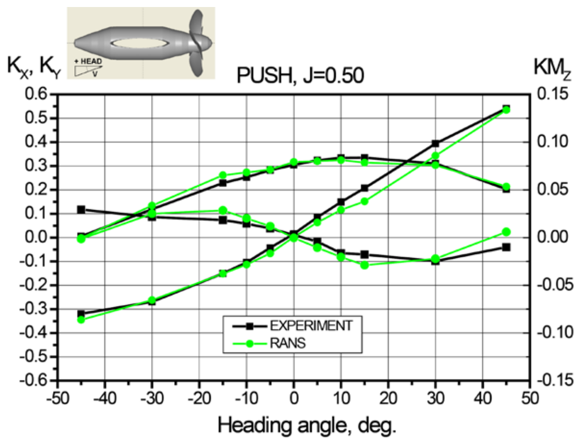


Figure 5: Comparison of computed and measured manoeuvring characteristics of a pushing thruster.

CFD modelling of propulsor-hull interaction

›› Senior Research Scientist Vladimir Krasilnikov,
›› Research Scientist Lucia Sileo

Interactions between propulsors and hull and between propulsor components involve phenomena which can only be investigated by employing advanced experimental techniques and cutting-edge computational methods. The simulation of propulsor-hull interactions is an ongoing activity in several of MARINTEK's R&D and commercial projects.

Sliding Mesh and Multi Reference Frame

Interaction phenomena can be modelled using RANS methods that adopt different approaches. When propeller performance has to be predicted or optimized, or the loads acting on blades and shaft are the focus of the study, the actual propeller geometry must be included in the numerical model. In such cases, either the Sliding Mesh (SM) approach or the Multi Reference Frame (MRF) representation can be used. In both cases the propeller is embedded in a sub-domain of the computational grid, which rotates in the SM procedure, whilst it is "fixed" in the MRF representation.

This second procedure requires fewer computational resources and time than the SM approach, and it is most suitable for the study of open-water propeller performance, with and without nozzle, when the position of the blades is not relevant to the main goal of the analysis.

Propeller-hull simulations that employ the SM method and include the effects of free surface by a Volume-of-Fraction (VoF) method are used in studies of self-propulsion characteristics of ships and effective inflow on propeller. MARINTEK recently performed such a study of the "benchmark" model of the KRISO container ship concept (Figure 1). The results show that the CFD method is capable of predicting accurately the characteristics of propeller in "behind" conditions and producing a qualitatively and quantitatively correct picture of the flow past a hull with an operating propeller (Table 1).

	KTB	KQB
Experiment	0.1703	0.0288
Calculation	0.1699	0.0292

Table 1: Comparison of measured and calculated "behind-hull" propeller characteristics under self-propulsion conditions. KRISO container ship, Fr=0.26, n=9.5[Hz].

Actuator disk approach

On the other hand, when a detailed description of the flow close to the propeller blades is not the principal aim of the study, or at a very early stage of a design process, the actuator disk representation can be used to model the effect of a working propeller under defined loading conditions.

The actuator disk concept has been widely used in RANS methods, especially for effective wake simulations, to evalu-

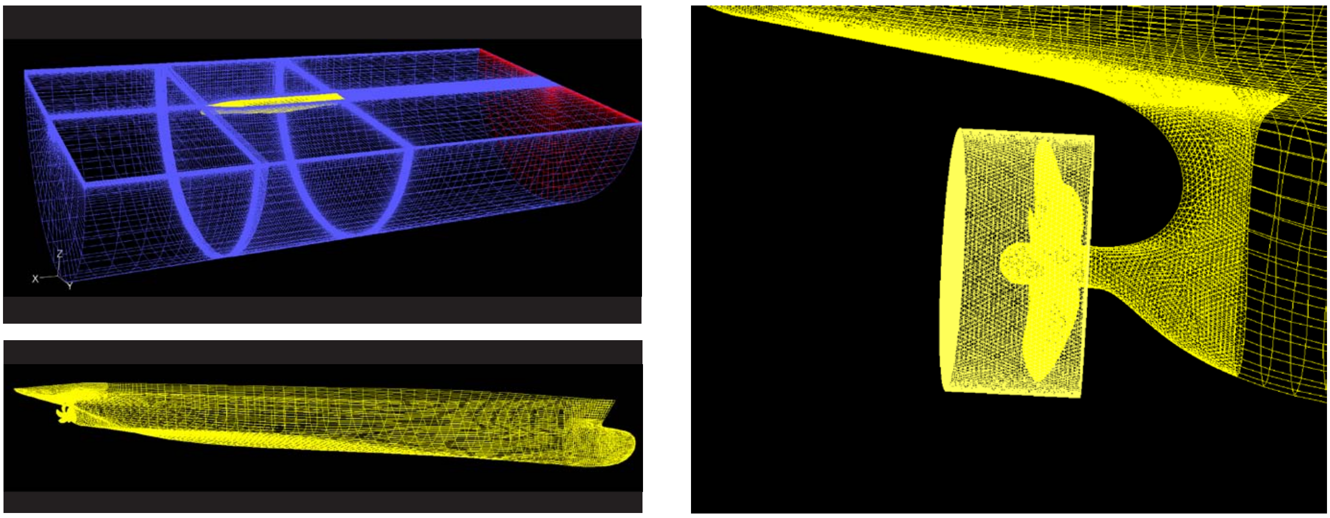


Figure 1: Details of the computational mesh used in self-propulsion simulations of the KRISO container ship.

ate the interaction between a propeller and a body immersed in its slipstream, and to predict the loading on nearby appendages, as shown in Figure 2.

The use of actuator disks instead of the actual geometry of the propellers offers significant savings in computational time and resources. An example of application of the actuator disk model is the study on the interaction between a stern tunnel thruster and twin ducted propellers. The whole ship was considered, and three propellers, including the ducts and the tunnel in the central skeg, were modelled using actuator disks. A strong interaction was predicted by the study, especially between the tunnel thruster and the propeller invested by its race. The jet from the tunnel is deflected and sucked in by the side propeller when it is working (Figure 2), and the suction field induced by the propeller strongly affects the pressure distribution in the area close to the tunnel exit, with a consequent loss of thrust from the tunnel thruster and deterioration in manoeuvrability.

The strong interaction identified by the numerical simulations was compared with the experimental measurements obtained during an extensive series of model tests carried out at MARINTEK. The use of the actuator disk concept also made it easier to deal with geometric modifications. In this study, in fact, the influence of the position of the tunnel on the interaction effect was studied, modifying the model geometry and moving the tunnel forward into five alternative positions.

For this study, the inclusion of the actual geometry of three

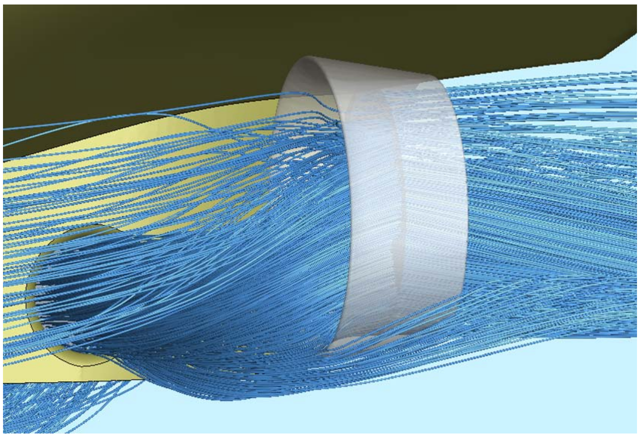


Figure 2: Interaction between a stern tunnel thruster and main propeller. Deflection of the jet from the thruster due to the action of the working propeller.

propellers would have been too costly, not only in terms of time and resources needed for calculations, but also in generating the computational mesh for all the different geometric configurations involved.

In self-propulsion simulations, the actuator disk with a certain distribution of momentum sources can be used in combination with potential flow methods, such as the panel method code AKPA, for propeller analysis. This approach is known as iterative viscous/potential coupled technique. Such a coupled solution becomes more challenging as soon as dynamic body simulations in either calm water or in waves are taken into consideration (Figure 3).

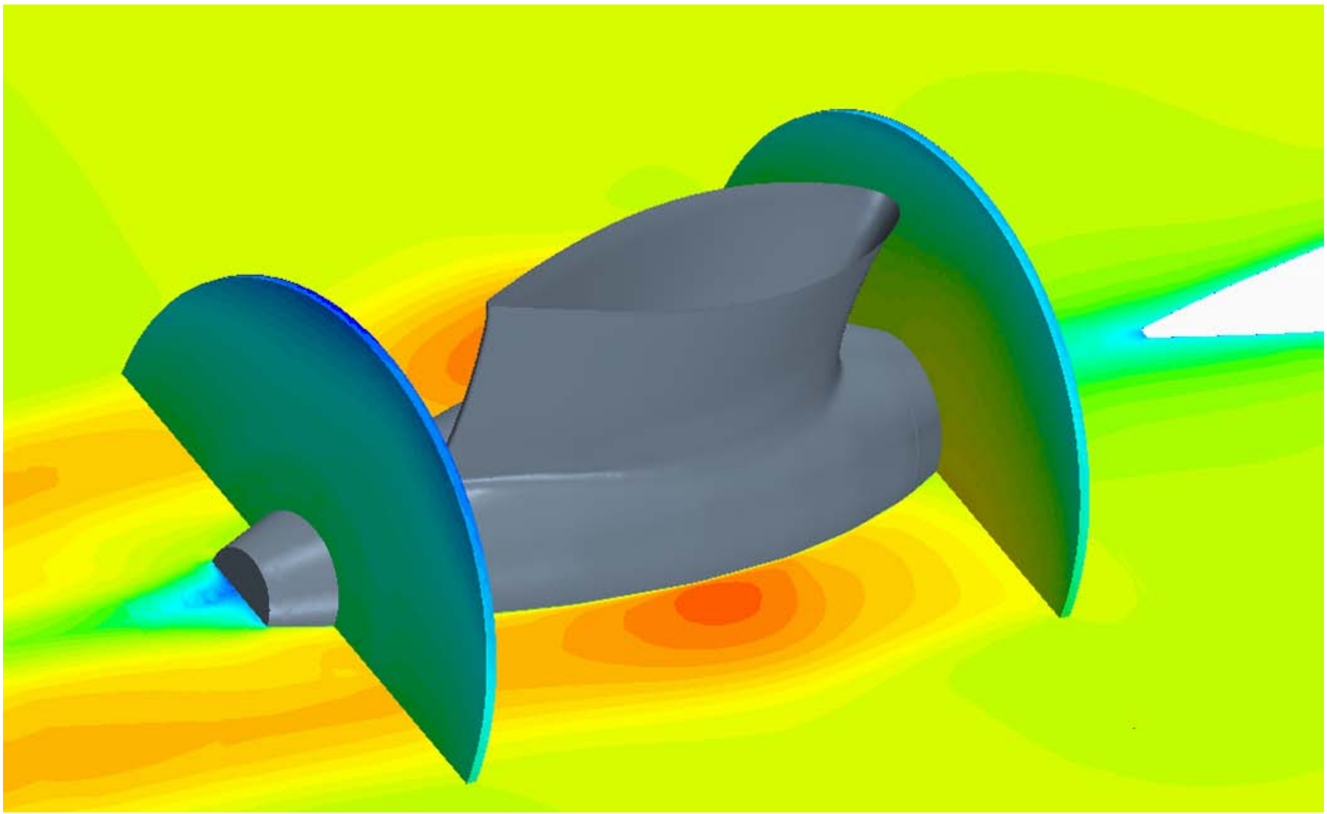


Figure 3: Momentum source in RANS methods used to model the action of a Schottel thruster behind a vessel. Coupling RANS-AKPA.

Wave-in-deck impacts

›› Research Scientist Csaba Pakozdi

The problem of wave-in-deck loading involves very complex physical mechanisms which demand close study, and experimental validation of the CFD setup used to examine this phenomenon is essential. MARINTEK's Wave Impact Load Joint Industry Project (JIP) focused on an idealised model test setup of a rectangular block in regular waves. The CFD setup used to reproduce this phenomenon was validated against the measurements from a 2D model test setup and a simple potential theory-based method.

For many existing large volume offshore platforms, the problem of insufficient air-gap clearance leading to wave slamming on their lower deck has become more acute of late. Indeed, some offshore structures are exposed to harsher weather conditions than allowed for in the original design, which explains the increased interest in this topic.

MARINTEK has performed 2D and 3D model test experiments of a block (fixed at a distance h above the calm water line) in regular waves. Comparisons with a simple potential theory-based method and against results from a CFD code (STAR-CCM+) were performed in terms of vertical and horizontal loading on the deck, as well as free-surface elevation.

For isolated impact events, a simplified approach, which does not consider the influence of one impact event on another, appears to be adequate to predict the vertical loading on the deck. However, results demonstrated that a second impact event closely following the first can have a much flatter free-surface profile as a result of its interaction with the diffracted wave from the first impact event, as shown in Figure 2. The impact of this flatter wave leads to a stronger water entry force which must be also taken into account in the design of the structure. A more complete CFD analysis resolving the diffracted wave is required to adequately predict the increase in vertical loading at the second impact.

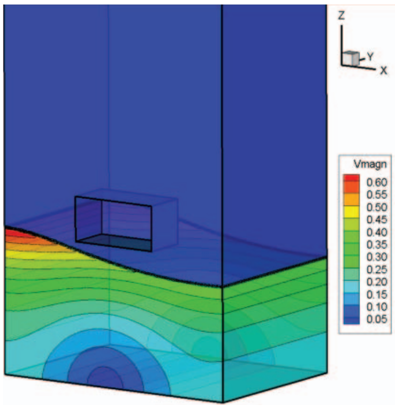


Figure 1: Velocity of the wave before the first impact event.

If there is a likelihood of steep wave groupings resulting in closely following wave-in-deck impact events, the simple method would therefore be non-conservative, and a CFD analysis or model test is advisable.

Comparison of CFD simulations with measurements shows a good match for the run-up and the impact load during the first impact event, while the horizontal loading of the second and third impact events was significantly under-predicted. This field is currently under study at MARINTEK and, in order to better reproduce the experimental conditions, the wave model is being improved.

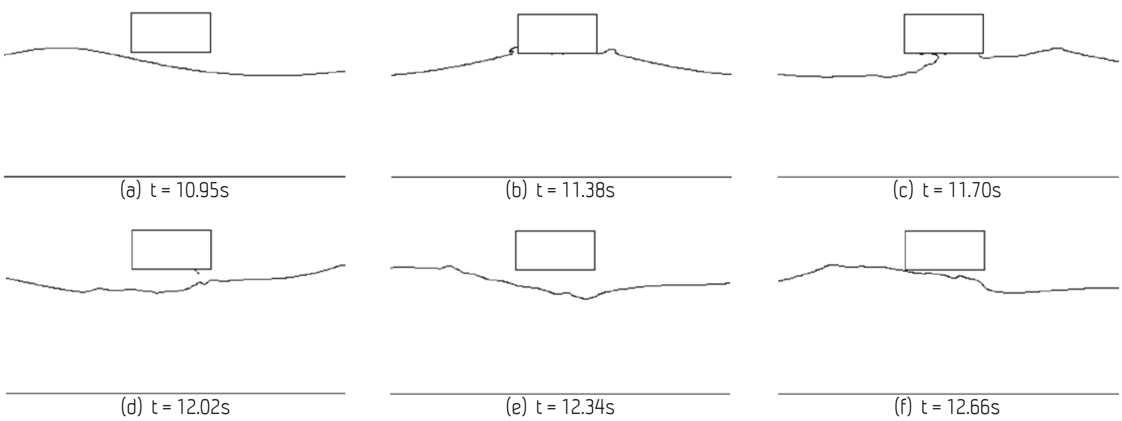


Figure 2: Snapshots of the free-surface profile before and after first impact, extracted from the CFD computation. The second impact has a much flatter free-surface profile as a result of its interaction with the diffracted wave from the first impact event.

Breaking wave impact on a platform column

›› Research Scientist Csaba Pakozdi

In order to gain greater insight into the phenomenon of breaking waves slamming on the legs of large-volume offshore platforms, MARINTEK's Wave Impact Load Joint Industry Project (JIP) has studied a long-crested breaking-wave impact on a rectangular cylinder with simplified deck structure.

The prediction of extreme wave impact loads on the columns of offshore structures is of major importance for the design and limit-state analysis of column-based offshore platforms such as semisubmersibles and gravity-base structures. To investigate this problem, one of the sub-tasks of MARINTEK's Wave Impact Loads JIP has focused on an idealised model test setup of a rectangular cylinder in breaking waves.

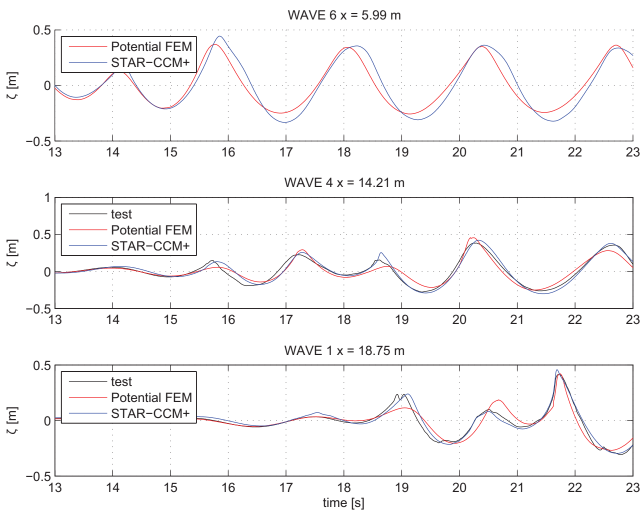


Figure 1: Time history of the calibration wave at three locations ahead from the breaking point.

The first challenge was to improve the wave boundary condition used in the CFD simulation in order to match the exact motion history of the wave-maker with the measured free-surface elevation throughout the computational domain. Among the simulation parameters specifically tuned to capture the physics of this simulation, the numerical diffusion of the wave propagation was significantly diminished by the use of a higher-order time-integration scheme for volume of fluid (VOF).

Figure 1 shows a comparison of the results obtained by CFD against the measurements. Details of the fluid velocity at the wave crest are presented in Figure 2.

The commercial CFD tool Star-CCM+ was used to reproduce the experiments. The model consists of a vertical column with a fragment of a horizontal deck attached. Although the horizontal loads on the cylinder were the focus for these experiments, a simplified horizontal deck structure was attached on top of the column both in order to obtain a more representative structure for fluid-structure interactions, and to facilitate wave-in-deck measurements, which is another important area of interest with regard to design loads.

The wave elevation obtained from the CFD simulation of a long-crested breaking wave and its impact on the cylinder and deck structure was compared to the free-surface elevation measurements.

An almost exact match was obtained between the computed wave profile and the measured wave profile, and spatially averaged slamming pressures look fairly similar to the model test observations.

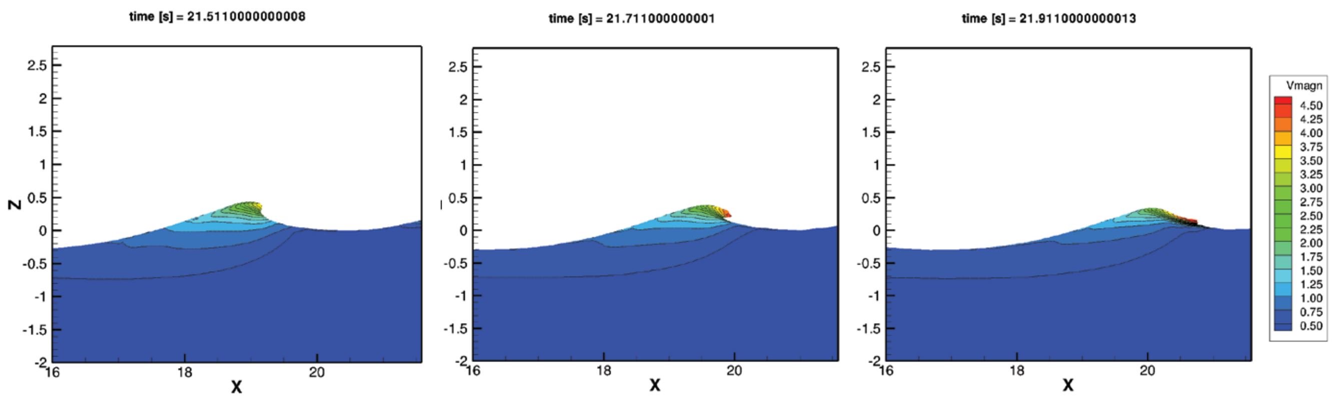


Figure 2: CFD simulation of the calibration wave at three different times.

Combined methodology for determination of wind and current coefficients

›› Research Scientist Thomas Sauder
›› Research Scientist Eloïse Croonenborghs

The use of a CFD model validated by a short, dedicated model test campaign has become MARINTEK's preferred approach for establishing accurate wind and current coefficients that are used in time-domain analysis of offshore structures. In addition to providing more insight into the involved physics, the validated CFD model enables us to obtain high-quality complementary data (damage conditions, alternative drafts ...) more rapidly and cheaply.

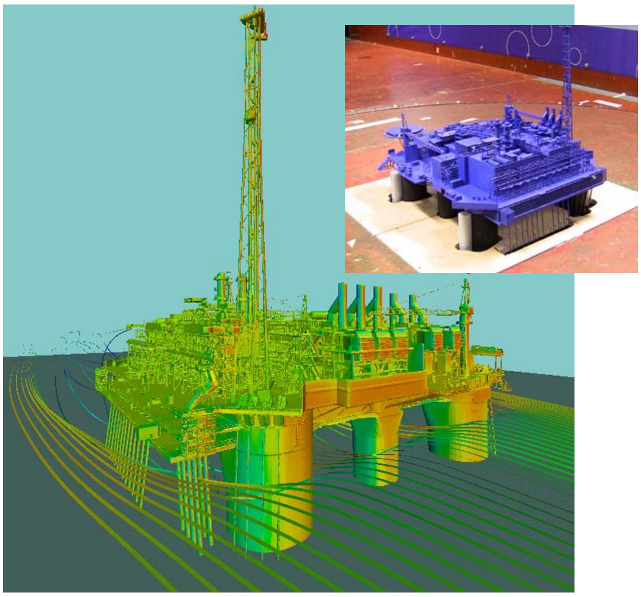


Figure 1: Pressure and streamlines on a semi-submersible platform. Comparison of the CFD model with the wind tunnel model.

As a complement to basin tests, time-domain simulation of offshore installations is an important way of verifying concepts under development. The SIMO software package developed at MARINTEK performs time-domain simulations in which current and wind loads are estimated on the basis of steady wind and current coefficients, as in many other similar programs. In the past, those coefficients were established either by wind tunnels or towing tests. Today, CFD is systematically becoming a part of the process.

First, a preliminary CFD analysis can, at a reduced cost, provide useful estimates for early dimensioning purposes, and allows insight into the underlying physics to be gained. At a

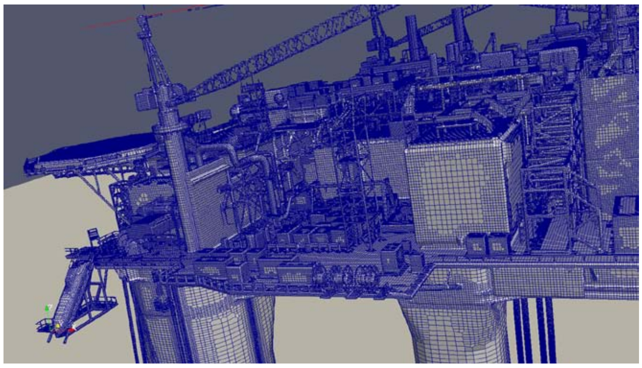


Figure 2: Detail of the surface mesh on a semi-submersible platform.

second stage, when the performance of a design needs to be accurately documented, a short experimental campaign can be used to validate the CFD model. The validated numerical model can then be used for the complete documentation of wind and current coefficients. Eventually, if deemed necessary at a later phase of the project, this flexible approach allows alternative setups that would be time-consuming and costly to re-test experimentally to be evaluated. Studies of damage conditions, of additional draught, or sensitivity studies of minor design parameters can, for instance, be performed using numerical methods alone.

As Figures 1 to 3 show, this approach has recently been used to study wind and current loads on a large semi-submersible. CFD analyses were performed using the OpenFOAM libraries. The wind study included a realistic atmospheric boundary layer profile of averaged wind velocity and turbulent kinetic energy. The size of the semi-submersible, together with the required level of detail, required a relatively large mesh of 13 million cells. Running the computations in parallel over thirty-two cores of our cluster enabled us to reach a very acceptable calculation time. The CFD model was compared with wind-tunnel tests performed under the operating conditions of the platform. Once it had been validated, the CFD models for wind and current loads were used for studies of a number of design variations.

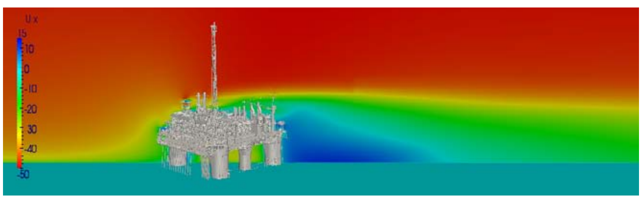


Figure 3: Velocity profile around a semi-submersible platform in oblique flow.

Analysis of gap resonance problems by a hybrid method

›› Research Scientist Thomas Sauder
›› Research Scientist Trygve Kristiansen

Gap problems refer to a class of hydrodynamic problems that include, for instance, the analysis of flow in moonpools, between ships in side-by-side configuration, and between a ship and a terminal. For all those problems, the coupled motions of the structure and of the entrapped fluid column, under various wave conditions, are decisive for operability assessments.

Classical potential theory seriously over-estimates ship and fluid motion under near-resonant conditions. This is due the absence of physical modelling of the flow separation and vortex shedding near sharp corners, which have a damping effect on the piston-like fluid motions in the gap. Examples of such sharp corners are bilge keels, damping plates, moonpool edges, or simply platform pontoons.

In the course of a three-year research project, MARINTEK performed numerical and experimental studies of this phenomenon. Three distinct numerical approaches were involved. The first was a fully nonlinear time-domain BEM code that included an inviscid vortex-tracking method. The vortex-tracking method follows the separated free-shear layer in time. This method provided accurate results but lacked robustness. Moreover, in practice the approach is limited to two-dimensional flow studies, although the formulation is valid also in three dimensions. The second method was a Finite Volume Method combined with a Volume of Fluid approach to capturing the interface. The main difficulty related to this method was that it was computationally resource-intensive. A third approach, which is currently under development, is a hybrid Finite Volume Method. Hybrid refers to the fact that it combines "potential" and "Navier-Stokes" domains, which

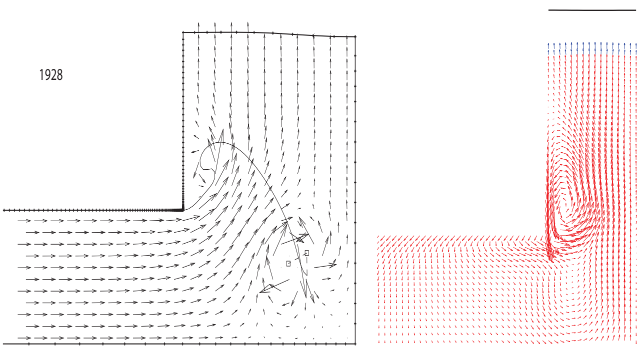


Figure 1: Snapshots from (a) BEM with vortex-tracking method (from [1]), (b) Hybrid method (from [2]).

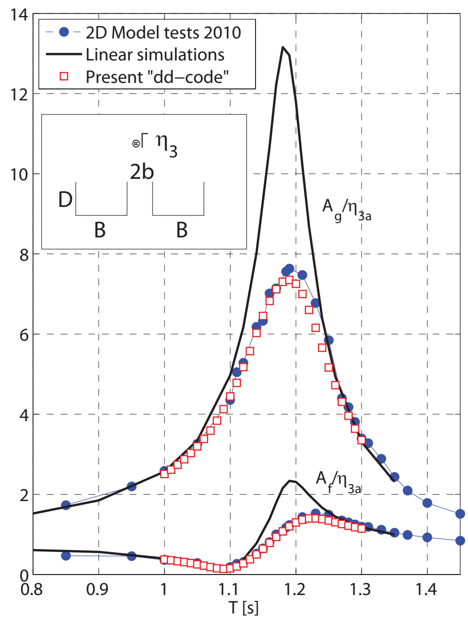


Figure 2: Validation results of hybrid method (from [2]). Forced heave amplitude is 5mm. «Present dd-code» means hybrid method. Linear simulations are calculated by the hybrid method with viscous domain turned off.

make it highly suitable for the study of resonance problems in which both waves and flow separation matter.

A snapshot from a moonpool simulation using the vortex-tracking method is shown in Figure 1(a). The box-shaped hull was forced to heave in sinusoidal motion at the natural frequency of its moonpool, causing a large piston-like water motion. The figure illustrates that the shed free-shear layer (vortex) from the sharp inlet edge caused a back-flow at the moonpool inlet, thus providing damping. A snapshot from a simulation using a two-dimensional hybrid code in a similar moonpool case is shown in Figure 1(b), where a similar vortex structure can be observed.

The results of a validation study of the hybrid method are shown in Figure 2. The agreement is good and the computational time required was low. One simulation that ran 30 wave periods with 80 time-steps per period (2400 time-steps) took only 73sec on a 2.4GHz PC.

References

[1] Kristiansen, T., Two-dimensional numerical and experimental studies of piston-mode resonance, PhD thesis, Norwegian University of Science and Technology (2009)
[2] Kristiansen T and Faltinsen, O. M., Gap resonance analyzed by a domain decomposition method, 26th Int. Workshop on Water Waves and Floating Bodies (2011)

Validation of an SPH sloshing simulation by experiments

›› Research Scientist Csaba Pakozdi

Sloshing is a violent fluid motion which is of great interest to the gas shipping industry. Among the tools used for sloshing analysis, numerical methods based on smooth particle hydrodynamics (SPH) are particularly promising for the analysis of violent fluid impacts. At MARINTEK, two-dimensional sloshing is being modelled with the aid of an in-house Sph code which was successfully validated by experimental and semi-analytical methods.

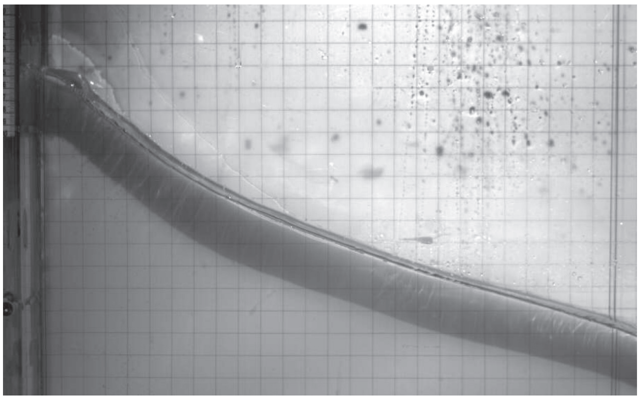


Figure 1: Snapshot from the high-speed video.

Fluid motion in containers has been investigated analytically, numerically and experimentally by many researchers for the past 50 years. Analytical solutions of linearised problems are valid for small oscillations with frequencies that are not close to the resonance frequencies of the fluid, but these have significant limitations. They cannot describe overturning waves, spray, run-up, roof impact or dry-bottom conditions.

MARINTEK uses the SPH method to model two-dimensional sloshing. This method has been successfully validated by experimental and semi-analytical methods. The SPH method is an interpolation method that allows any function to be expressed in terms of its values at a set of disordered points - the particles. The standard SPH formulation was modified to focus on implementation of the Verlet time integration scheme,which enables long-duration sloshing simulations to be performed, due to its good momentum- and energy-conservation properties.

Since model tests are the most accurate method for analysing sloshing, a series of tests was performed in a two-dimensional tank. A regular one-degree-of-freedom motion with small amplitude was imposed for various frequencies around the fluid's natural frequency and for three filling levels, 17, 33 and 40%, of the tank.

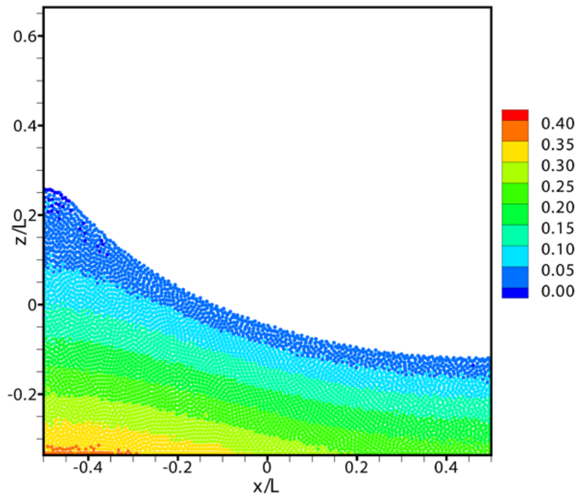


Figure 2: Non-dimensional pressure obtained by SPH computations.

The SPH simulations were then performed for the same conditions as had been tested experimentally. The acceleration signal measured on the tank was input to the SPH simulations. The SPH code was validated in terms of pressure and free-surface elevation. Figure 2 shows the pressure distribution obtained by SPH under the condition and at the moment in time of the snapshot shown in Figure 1. The time series of the pressure was recorded by means of sensors mounted on the sides of the tank. Figure 3 shows the evolution of pressure at the site of one of these probes.

Good agreement with the experimental data in terms of free surface elevation and pressure was found for a number of specific cases. This shows that use of the SPH method for regular, non-impact flows with carefully chosen coefficients, leads to accurate reproduction of such a sensitive phenomenon as sloshing.

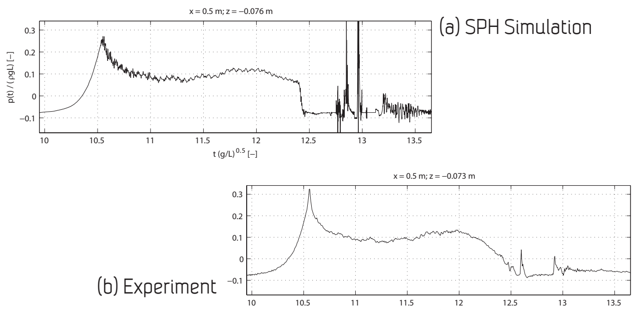


Figure 3: Time history of the pressure at a single measurement point.

Use of complex inlet boundary conditions for accelerated studies of green-water events

›› Research Scientist Csaba Pakozdi

Model tests have traditionally played an important role in estimating hydrodynamic loads during "green-water events" (i.e. when ocean waves break onto a ship's deck or superstructure). To be able to predict wave-loads correctly, it is important that the characteristics of the incoming wave are realistic [1,2].

This project adopts a smoothed-particle hydrodynamics (SPH) approach to modeling green-water events, based on a complex velocity inlet boundary condition. In naval architecture, practical hydrodynamic engineering tools have been developed based on both experimental data and nonlinear random-wave modeling, including 3D linear diffraction theory, which allows realistic waveforms for green-water events to be estimated. We use one such tool, Kinema2 [1], for the purpose of describing spatial and temporal variations in the wave as it breaks along the deck perimeter.

This approach greatly reduces the number of degrees of freedom (SPH particles) needed to solve the problem, thus rendering the analysis feasible on a laptop computer. We compare the SPH results for the resulting water elevation and velocity on deck with CFD and experimental time series from model tests.

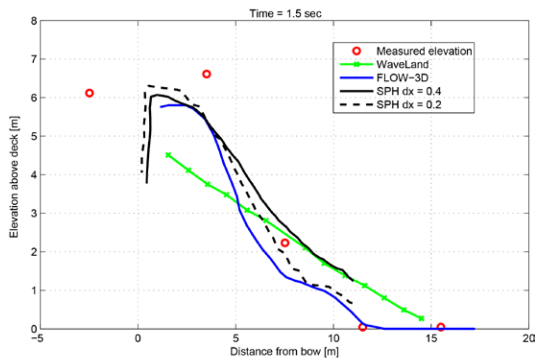


Figure 2: Water height on the deck; comparison measurement with predictions of several numerical methods.

The results of the numerical simulation show that the SPH method is able to predict the water propagation on the deck with the same accuracy as a Volume of Fluid CFD method which was more computationnally expensive [3].

References

[1] Stansberg, C.T. et.al., "Simple tool for prediction of green-water and bow-flare slamming on FPSO, OMAE 2009-79489

[2] Gómez-Gesteira, M. et.al., "Green-water overtopping analyzed with a SPH model. Ocean Engineering 32 (2005) 223-238.

[3] Pakozdi, C. et.al., "Using a simplified Smoothed Particle Hydrodynamics model to simulate green water on the deck, OMAE 2012--83847.

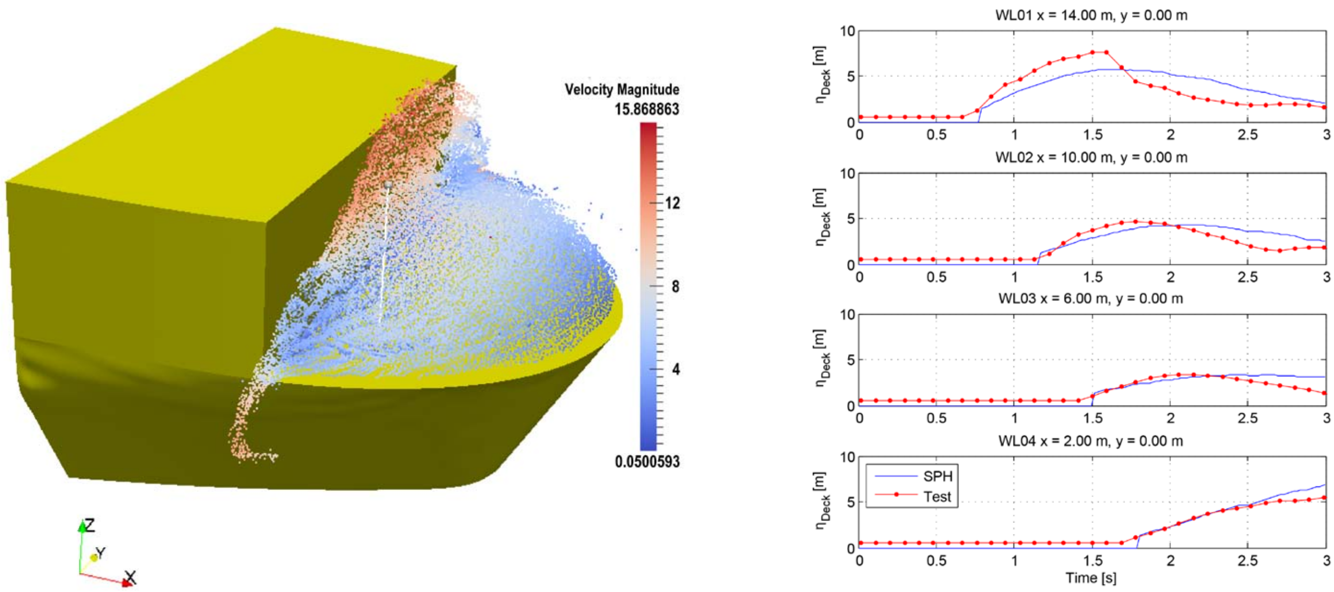


Figure 1: Visualization of green water event (left) water elevation along centre line (right).

CFD capabilities and hardware at MARINTEK

Computational Fluid Dynamics is a steadily expanding part of our expertise in analyzing and optimizing vessels and offshore structures. The conjunction of solid competence, efficient software and reliable hardware is an essential factor that enables us to deliver high-quality expertise.

The MARINTEK CFD team has the qualifications and experience needed to understand the underlying physics of a wide range of problems in marine technology. Our world-class laboratories enable us to validate CFD tools, evaluate their limitations and develop them further.

MARINTEK has performed research projects for Norwegian and international clients using a wide range of CFD solvers, namely Fluent, StarCCM+, FINE/Marine and OpenFOAM. Our philosophy is to use the most efficient tool on a case-by-case basis in order to provide high-quality CFD analyses for the whole range of challenges that face the maritime industry.

CFD analyses can require weeks of execution time on a single CPU, and in practice, parallelization over several CPU's may be the only way of achieving results for large models. MARINTEK operates a Linux cluster that comprises 256 64-bit cores, 2 GB RAM/core, and an InfiniBand network architecture. This configuration enables us to obtain maximum performance with CFD codes that have been designed and optimized for cluster use. Separate work-stations with state-of-the-art graphical capabilities are used for grid generation and visualisation of results.

Authors

Eloïse Croonenborghs
Research scientist
Phone: +47 926 78 061
E-mail: Eloise.Croonenborghs@marintek.sintef.no

Lucia Sileo
Research scientist
Phone: +47 467 48 082
E-mail: Lucia.Sileo@marintek.sintef.no

Anders Östman
Research scientist
Phone: +47 482 21 867
E-mail: Anders.Ostman@marintek.sintef.no

Vladimir Krasilnikov
Senior research scientist
Phone: +47 920 90 884
E-mail: Vladimir.Krasilnikov@marintek.sintef.no

Csaba Pakozdi
Research scientist
Phone: +47 454 27 783
E-mail: Csaba.Pakozdi@marintek.sintef.no

Thomas Sauder
Research scientist
Phone: +47 452 76 069
E-mail: Thomas.Sauder@marintek.sintef.no

Trygve Kristiansen
Research scientist
Phone: +47 917 02 628
E-mail: Trygve.Kristiansen@marintek.sintef.no

MARINTEK
Norwegian Marine Technology Research Institute

Otto Nielsens veg 10, P.O.Box 4125 Valentinlyst
NO-7450 Trondheim, Norway

Phone: +47 7359 5500 Fax: +47 7359 5776
E-mail: marintek@marintek.sintef.no
www.marintek.no

SINTEF