Hydrodynamics – Highlights

Our research tools include experiments, analytical methods, Computational Fluid Dynamics (CFD) and approximate numerical methods. Our research is relevant to sea transport, marine operations, aquaculture plant design and Very Large Floating Structures (VLFSs).

Prof. Odd M. Faltinsen, Head of Hydrodynamics

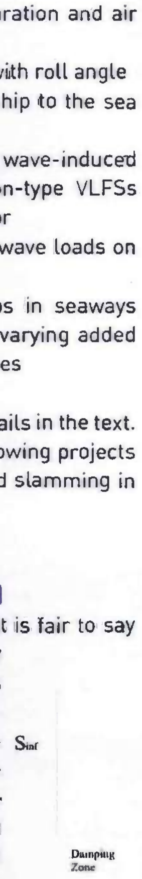
Main research activities

- Broaching in waves
- Broaching of semi-displacement vessels in calm water
- Numerical study of the nonlinear steady wave problem of semi-displacement and planing vessels
- Freshwater filled fabric structures in waves
- Replenishment operations of two ships
- Water entry with non-viscous flow separation and air cavity formation
- Water entry of a bow-flare ship section with roll angle
- Lowering of porous structures from a ship to the sea floor
- Hydroelastic bottom slamming and wave-induced global hydroelastic behavior of pontoon-type VLFSs including the effect of an uneven sea floor
- Numerical and experimental studies of wave loads on aquaculture plant floaters
- Calculations of the speed loss of ships in seaways accounting for vessel dynamics, slowly varying added resistance and propulsion change in waves

Other activities are presented in greater details in the text. INSEAN is a cooperating partner in the following projects on greenwater events and sloshing-induced slamming in shallow-liquid conditions.

Computational Fluid Dynamics (CFD)

A broad variety of CFD methods exist and it is fair to say that there is no unique method. Our policy has been to develop numerical methods from scratch and to strongly emphasize verification and validation. Our methods include the Finite Difference Method (FDM), the Constrained Interpolation (CIP) method, the Finite Volume Method (FVM), the Smoothed Particle Hydrodynamics (SPH) method, and the Boundary Element Method (BEM). Within the Eulerian field solvers colour function techniques have been used as free-surface tracking methods, for example, the Level-Set (LS) and Volume of Fluid (VOF) methods.



From a computational point of view, the application of a CFD method to solve wave-induced motions and loads on structures in a seaway, and associated probability density functions for the structural response, is demanding. Domain decomposition techniques that combine numerical solvers for different fluid domains can be an efficient way to reduce computational time. When the angle between the impacting free surface and the body surface is small, it is challenging to properly describe the slamming loads by a CFD method. Our experience is that a BEM in combination with a local analytical solution may provide the best estimates.

Weakly nonlinear wave-body problems

Examples of practical applications are ship springing, gravity-based platform ringing, slowdrift oscillations of moored floating structures and nonlinear wave run-up along structures. A traditional perturbation method used in engineering applications is based on an inertial coordinate system, but this method fails if the oscillating body has sharp corners and causes numerical difficulties with high body curvature. A newly developed numerical method using a body-fixed coordinate system near the ship or floating ocean structure copes with such difficulties. A domain decomposition technique is applied and inner and outer domains are defined (see Fig. H1). A high-order Boundary Element Method, based on cubic shape functions, is used in the studies.

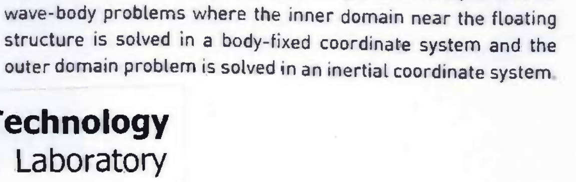


Fig. H1: Domain decomposition technique for weakly nonlinear wave-body problems where the inner domain near the floating structure is solved in a body-fixed coordinate system and the outer domain problem is solved in an inertial coordinate system.

Delft University of Technology
Ship Hydromechanics Laboratory
Library
Mekelweg 2 2628 CD Delft
Phone: +31 (0)15 2786873
E-mail: p.w.deheer@tudelft.nl

Green water events

Last year, a preliminary 3D Domain-Decomposition (DD) strategy was developed, coupling a weakly-nonlinear seakeeping solver with a shallow-water approximation for the in-deck problem. It is efficient and globally reliable, but unable to recover all the nonlinearities: the water run-up is not properly described when high-speed jets are formed and the initial plunging-wave phase is not handled (see examples in Fig. H2).

Fig. H2: Water-on-deck experiments: high-speed jet rising at the bow (left) and initial wave-plunging phase during water shipping (right).

To fully handle the nonlinear effects, a 3D Navier-Stokes (NS) Level-Set (LS) method using a FDM will be applied in a sea region containing the vessel-upstream portion to handle water-on-deck occurrence. Initial and boundary conditions from a linear seakeeping analysis are the input to the NS solver, and predicted greenwater loads are the output to correct the ship motions otherwise based on a linear potential theory. The first step included developing the field solver and assessing its robustness and reliability. Figure H3 shows the NS-LS verification in terms of wave elevation, pressure and speed for a pure wave-propagation problem: an airy wave oblique to the grid axes which is numerically challenging.

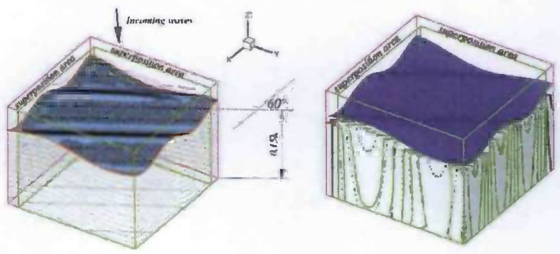


Fig. H3: Airy wave with steepness kA=0.03. Left: theoretical (meshed in black) and numerical (blue shaded) free surface after two wave periods. Pressure contours are plotted on the side of the domain. Right: theoretical (black) and numerical (green) contour plots of x-velocity component.

A hybrid algorithm models the vessel (see example in Fig. H4 left): the LS function identifies the ship (contour lines around the initial ship location) and solid particles are convected in time and used to enforce the body-boundary conditions on the grid (the grey isosurface is the new ship position). The resulting method can handle extreme conditions with portions of the vehicle leaving the inner domain. A forced-pitch example is shown to the right in Figure H4.

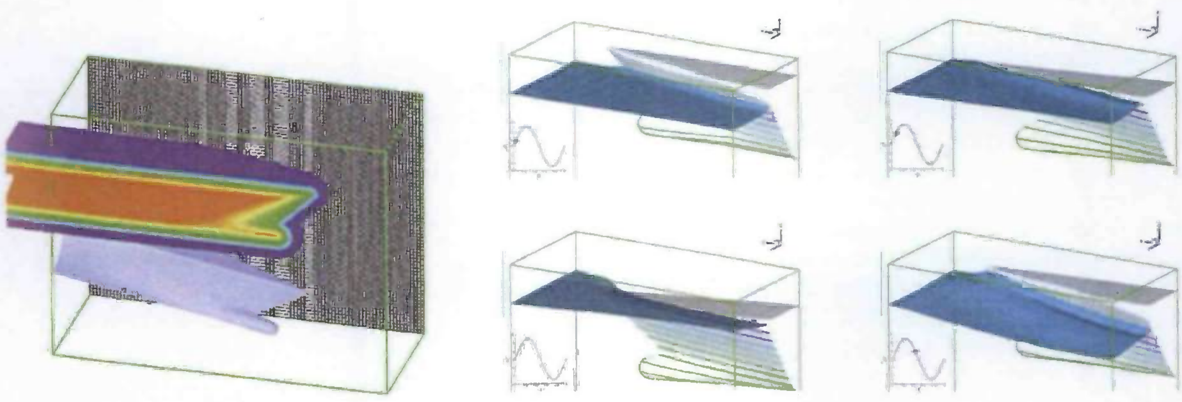


Fig. H4: Left: example of hybrid tracking of the body position. A set of particles, labelled with their distance function from the hull surface, are advected on the grid. Right: free-surface and pressure contour plots around the bow of the pitch-forced hull. The actual pitch angle is indicated by the black point on the pitch curve in red.

Sloshing

Analytically-based solutions for linear sloshing have been derived based on a variational formulation and analytical continuation. Formulas accounting for interior small-volume tank structures have also been developed. A simplified and very time-efficient analytically-based linear modal method, that gives approximate predictions of transient sloshing in two-dimensional circular and spherical tanks, has been derived. Comparisons with published experimental results for circular tanks are encouraging.

An accurate linear modal method for sloshing in a two-dimensional circular tank has been developed. Description of the sloshing behaviour for filling depths larger than the radius requires accounting for the flow singularities at the intersection between the mean free surface and the body surface.

Fig. H5: The two-dimensional sloshing experiments were conducted in a tank with a perforated wall in the centre.

Theoretical and experimental studies relevant for swash bulkheads have been performed. The optimal ratio between the opening area and the projected area of the swash bulkhead that minimises sloshing has been studied as a function of tank excitation (forcing frequencies and amplitudes) and filling depth. The optimal area ratios are larger than the recommended practice, and generally increase with lateral tank excitation amplitude. The physics is related to a decrease in the lowest natural sloshing

A Numerical Study of a Damaged Ship in Beam Sea Waves

Xiangjun Kong successfully defended his PhD thesis on 18 December 2009

Supervisor: Professor Odd M. Faltinsen

Current position: Senior Technical Professional – Marine, Gothenburg

The objective of this study was to estimate the time-to-capsizing of a given damaged ship. Potential flow theory was used to study ship motions in regular beam sea waves with ingress/egress flooding and/or sloshing effects considered.

Milestones on the research path were the efficient numerical technique of weighted average flux to solve the shallow water equations, multimodal method to solve the sloshing problem, and proper geometry model Hull Reshaped Method to study the damaged ship motions with submerged opening.

To study the behaviour of a damaged ship in waves it is crucial to follow three considerations. The first is that the ship motion equations used in the simulations have six degrees-of-freedom in order to represent realistic ship capsizing dynamics. The second is to correctly model the flooding flow through the openings. The final aspect is the three dimensional simulation of the floodwater flow on the deck or in the compartment, and the prediction of induced loads on the damaged ships since the flooding flow cannot generally be adequately described by a two-dimensional model.

In addition to the natural roll resonance, the piston mode and sloshing resonances were numerically observed. By applying simplified theoretical analysis, these two resonances were further confirmed for a damaged ship with an opening in the hull defined by the SOLAS rule. The resulting physical problems, for instance, dry bottom, roof impact and possible structural failure in the damaged compartment, were predicted from the simulations.

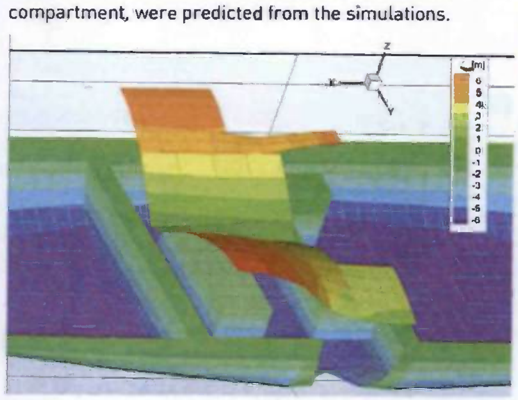


Fig. H6: Resonant wave amplitude in the damaged compartment of the ITTC ship at sloshing resonance ω =0.90rad/s. and wave height H_w=1.2m.

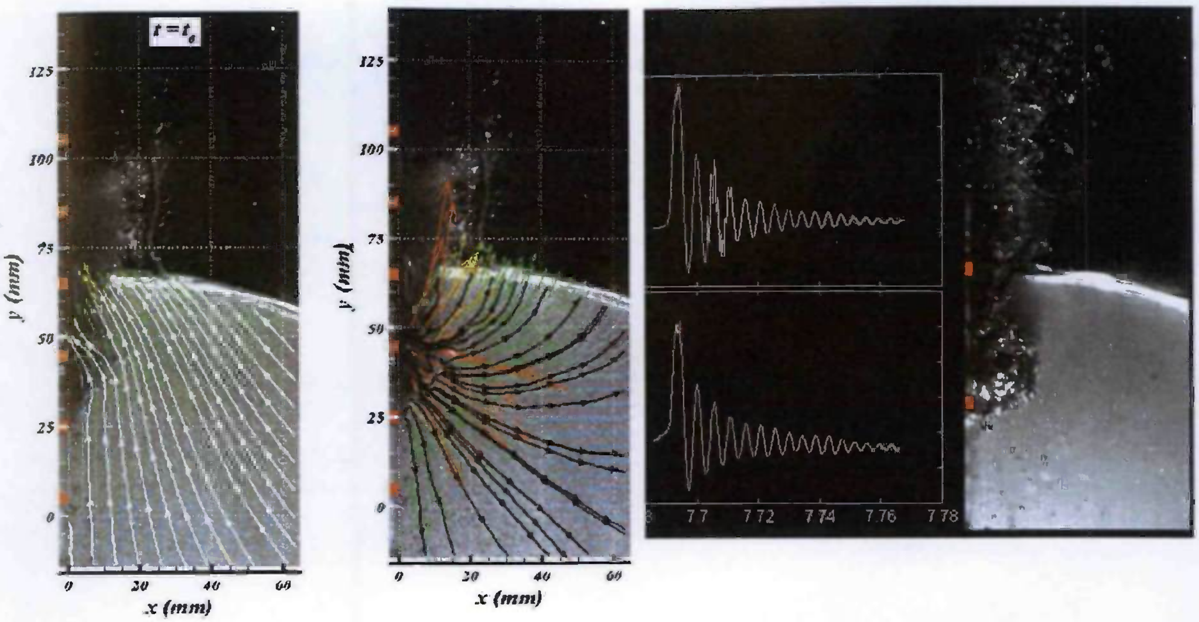


Fig. H7: Air-cavity entrapment during water-wall impact. Streamlines (left) and velocity field (centre) at the impact time t_0; $p_0=p_a$. Right: pressure analysis at two wall locations (red markers) during impact; $p_0=0.75p_a$. $p_0=0.75p_a$. p_0 and p_a are ambient (ullage) and atmospheric pressure.

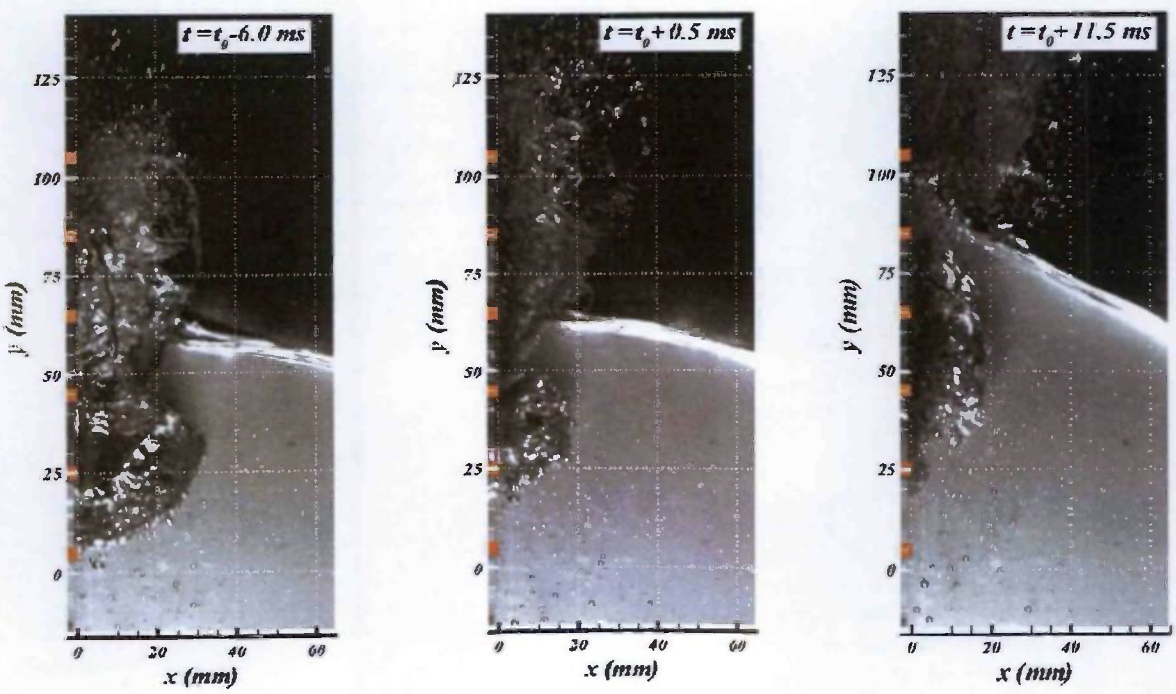
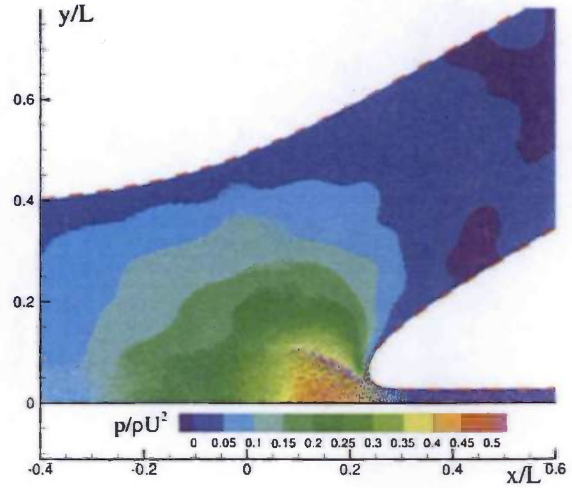
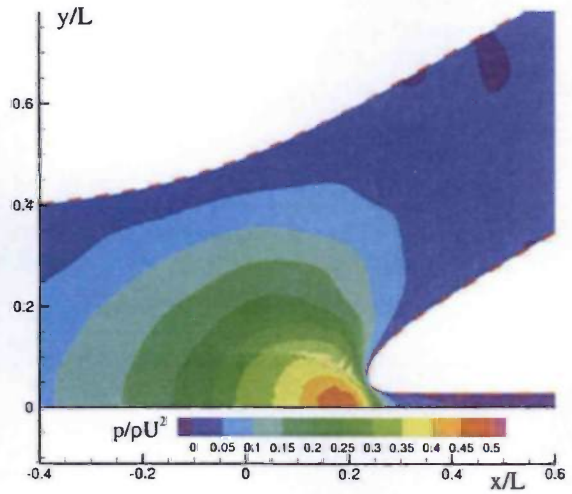


Fig. H8: Stages 1 (left), 2 (centre) and 3 (right); $p_0=0.75p_a$. P_0 and p_a are ambient (ullage) and atmospheric pressure.





period and damping due to cross-flow in the opening of a swash bulkhead. Figure H5 gives a picture from the model set-up. The tests assume a two-dimensional flow; a perforated wall is placed in the middle of the tank. The sloshing in each part of the tank behaves differently indicating that different solution branches are possible for steady-state nonlinear sloshing.

Gas cavity formation and resulting pressure oscillations during tank roof impact have been studied experimentally and numerically. The nonlinear liquid flow is coupled with the nonlinear one-dimensional gas flow before the closure of the gas cavity and is solved by a BEM. The effect of gas flow compressibility is investigated and matters in the final stage before closure. The gas flow predictions show a singular behavior at closure and are believed to be analogous to the "water hammer" problem associated with the closure of a pipe flow. The pressure in the gas cushion is assumed spatially constant after closure and is coupled with the liquid flow. Reasons for the experimental findings of temporal decay in the gas cavity oscillation amplitude have been experimentally and numerically investigated. Gas cavity leakage and changes in liquid flow are important factors. However, additional causes for the decay have not been identified.

A systematic experimental study on wave-impact events in shallow-liquid conditions associated with sloshing in depressurised 2D tanks has addressed the role of ambient (ullage) pressure. The impact process during the cavity formation, its evolution and its collapse into a mixture of water and air bubbles, was analysed in terms of the kinematic and dynamic features of the flow. Some examples of the investigated variables are shown in Figure H7.

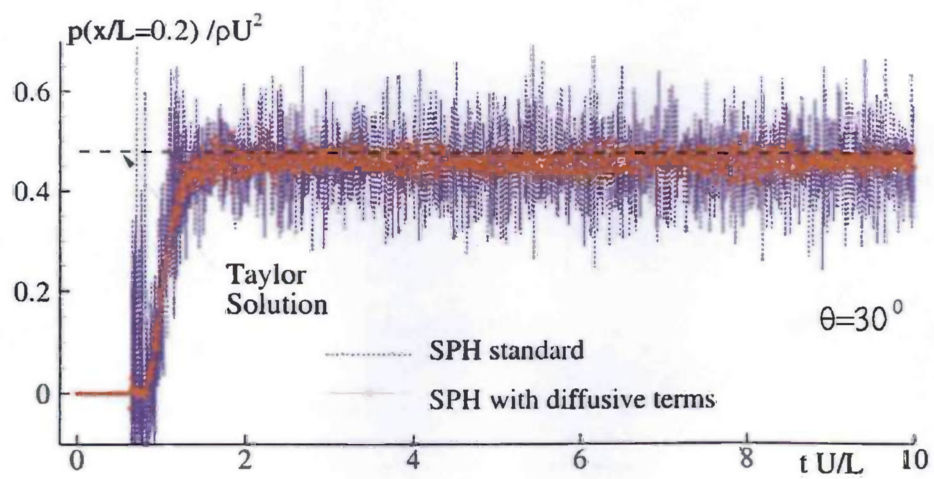


Fig. H9: 2D jet impinging on a flat plate with $\theta = \pi/6$. Pressure field by the standard (top) and present (centre) SPH and their pressure evolutions at the initial-impact location (bottom), respectively, in dotted line and line with symbols. The analytical free-surface and pressure solutions are reported as dashed lines.

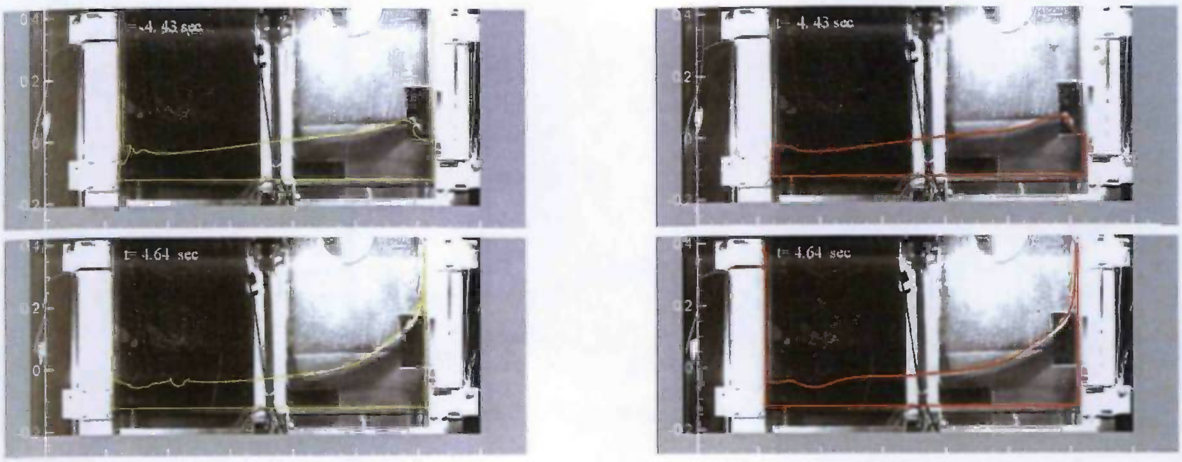


Fig. H10: Flip-through event during sloshing. SPH (left) and NS-LS (right) free-surface snapshots. Numerical results are compared against experiments.

Four stages characterise the flow evolution:
1) the cavity closure,
2) its isotropic compression/expansion,
3) its anisotropic compression/expansion and
4) its raise along the wall.
The first two stages are governed mainly by the airleakage, the last two by the surrounding hydrodynamic flow which contributes to squeezing the bubble horizontally and to convecting it up along the wall. The first three stages are shown in Figure H8.

Additional physical effects have been introduced in the Smoothed Particle Hydrodynamics (SPH) model developed at INSEAN in the past years. Now, viscous and surface tension effects as well as multiphase flows are described correctly and accurately. A new formulation addresses the SPH challenges in handling local loads during impact: a diffusive term in the continuity equation allows mass exchange among particles but preserves the total mass, this reduces the noise affecting standard-SPH impact pressures (Fig. H9); the energy conservation equation

A Numerical Study of Wave-in-Deck Impact using a 2D Constrained Interpolation Profile Method

Tone M. Vestbøstad successfully defended her PhD thesis on 31 August 2009.
Supervisor: Prof. Odd M. Faltinsen
Current position: Principal Engineer, Statoil

This thesis began with the objective of studying the flow around violent wave impacts on offshore platforms. The CIP method was chosen as the numerical model.
The main findings can be summarised as follows:

- Coding and careful verification and validation of a 2D CIP method
- Comprehensive comparison of four different CIP based interface capturing methods
- Numerical study of the wave impact process using a numerical wave tank
- Comparison of numerical and experimental results for the wave impact process, including evaluation of the uncertainties in both the numerical method and the model tests.


Fig. H11: Snapshots from wave impact experiment. Photo: Rolf Baarholm/Trygve Kristiansen.

is solved, this non-isentropic pressure variation during the impact thus reducing computational costs.

Figure H10 documents the SPH and NS-LS application to a 2D sloshing problem during a flip-through event. The water evolution fits the experimental data. The wall pressure in the impact area (see Fig. H12) is consistent with the measurements, the NS-LS anticipation of the event is due to a lower resolution used in the simulation.

Procedures for evaluating sloshing-induced slamming in prismatic membrane LNG tanks have been studied. This includes how to scale model test results of slamming, and the necessity to account for nonlinear global sloshing loads when numerically predicting the mutual interaction between sloshing and ship motions in a seaway.

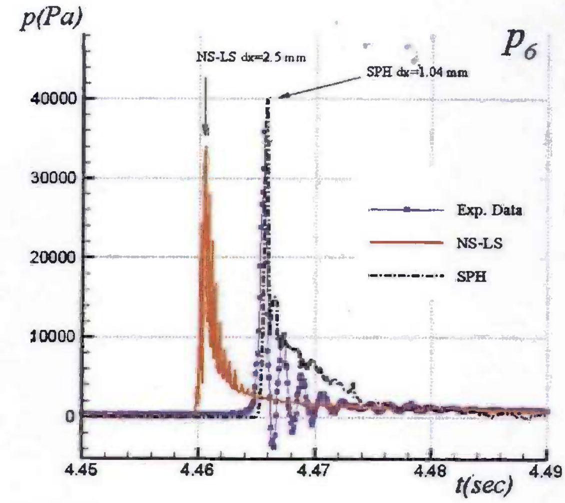


Fig. H12: Pressure evolution at wall location near the impact impact during the flip-through event shown in Figure H10.

Two-dimensional Numerical and Experimental Studies of Piston-mode Resonance

Trygve Kristiansen successfully defended his PhD thesis on 1 April 2009.
Supervisor: Professor Odd M. Faltinsen
Current position: Researcher at CeSOS

This thesis focused on resonant piston-mode motion in two dimensions. Piston-mode fluid motion is the near vertical, massive fluid flow in a moonpool or in between a ship and a terminal when the ship is moored alongside the terminal. This type of resonance is also called gap resonance. At resonance, large amplitude and massive fluid motion will occur. This may be problematic in practice. For example, resonant piston-mode motion in a moonpool may cause equipment damage due to slamming and may also prevent normal operation. Another example is that resonant piston-mode motion induces large wave-frequency forces as well as large drift forces on a ship moored alongside a terminal.

Standard engineering tools are based mostly on linear theory. Linear theory typically overpredicts the piston-mode amplitude at resonance from physical experiments by several times. It was therefore of interest to understand more about the nature of these gap resonance phenomena.

Our study revealed that viscous flow separation from the bilge keels of the ship has a significant damping effect on the resonant piston-mode motion. We studied the problem by conducting physical experiments and by a nonlinear numerical wave tank which was based on a Boundary Element Method (BEM) and including a vortex tracking method. The user could decide whether to include flow separation or not by a flag in the program input file. This way, we could compare nonlinear simulations with and without flow separation against linear theory and against experiments. We concluded that flow separation from the bilge keels was the cause of the discrepancies, and that the effects from the nonlinear free surface conditions were small.

Fig. H13: The liquefied natural gas (LNG) carrier, Dukhan, arrives at the Adriatic LNG Terminal. Photo: Business Wire/Exxon Mobil.