SUMMARY

Green water on deck, sloshing, slamming, high speed marine vehicles and Computational Fluid Dynamics (CFD) are selected as main topics. Slamming involves many physical effects. When analyzing slamming, one must always have the structural reaction in mind. This necessitates that hydroelastic effects are considered. The many hydrodynamic phenomena for the three main categories of high-speed vessels, i.e., vessels supported by the hull, foils and air cushions are considered. Dynamic instabilities, cavitation and ventilation are limiting factors for their performance. The coupling with automatic control is discussed. A brief overview of the many different CFD methods is given and advantages and disadvantages are discussed.

1. INTRODUCTION

There is a broad area of marine structures needed for ocean transport, exploitation of subsea hydrocarbons and wave energy, sea food production as well as for marine infrastructure. Challenging hydrodynamic aspects are focused on, in particular for violent fluid motion and high-speed vessels. However, hydrodynamics must be linked to other disciplines such as structural mechanics and automatic control. There is a general tendency that specialists in marine hydrodynamics work on separate disciplines such as resistance, propulsion, manoeuvring and seakeeping without always combining the knowledge.

Examples on violent fluid motions are green water on deck, sloshing and slamming. Capsizing of damaged ship with water ingress / egress in waves has similarities both with sloshing in tanks and green water on deck. Sloshing in moon pools is another example that could be mentioned. An important issue is to design efficient damping devices, such as perforated walls in the pool. A similar resonance phenomenon occurs between two adjacent ships in waves, or for a ship alongside a terminal. Conventional engineering tools based on linear panel methods cannot accurately describe the flow.

2. GREEN WATER ON DECK

There is extensive work worldwide in applying Computational Fluid Dynamics (CFD) to green water on deck. However, most work is related to two-dimensional flow. An attempt has been made by Greco et al. (2007) to classify how the different green water phenomena occur as a function of wave parameters in head sea conditions for stationary ships with blunt bows, based on experimental and theoretical studies of a restrained two-dimensional body. A typical application is to a Floating Production Storage and Offloading (FPSO) vessel. Interaction between the incident waves and the hull plays an important role. One type of green water is the so called dam-breaking phenomenon, where a vertical wall of water is generated at the edge of the deck due to the relative vertical motion between the ship and the waves. The subsequent motion of the water resembles the breaking of a dam with water flooding at high speed (15-20 m/s) along the deck. Its impact against deck structures and equipment can cause serious damage.

A second scenario is water hitting the deck as a plunging breaker. Actually, the plunging breaker may hit a deck house in the forward part of the ship. A third case is the "hammer fist" effect of water on deck, using an analogy to karate. A large mass of water rises above the
W/W ratio. Here W is the maximum vertical velocity of the incident waves and W is the maximum vertical fluid velocity at the ship's bow.

Wave-body interaction plays a similar decisive role in bow stem slamming as in the green-water situation.

3. SLOSHING

Sloshing in a partially filled container is a violent resonant free surface flow with strongly nonlinear behaviour. A partially filled ship tank will experience violent fluid motion when the ship motions contain energy in the vicinity of the highest natural period for the liquid motion inside the tank. Liquid impact on the tank roof is then likely to occur for larger filling ratios. The consequence is wave breaking, spray and mixing of gas and liquid. Actually, extreme cases with gas bubbles everywhere in the liquid have been experimentally observed.

Because sloshing is a typical resonance phenomenon, it is not necessarily the most extreme ship motions or external wave loads that cause the most severe sloshing. This implies that external wave induced loads can in many practical cases be described by linear theory.

The ship motions excite sloshing, which in return affects the ship motions. Ships equipped with anti-rolling tanks utilize this effect. The sloshing induced roll moment on the vessel will cause roll damping by properly choosing the highest natural sloshing period close to the roll natural period. FPSO units sometimes have several partially filled tanks during operation. The wave induced motions and loads on these ships will then be influenced by the dynamic motion of the liquid in the tanks. Because ship motions can strongly affect the mean and slowly-varying wave drift forces and moments, sloshing may also matter in a station-keeping analysis.

The tank shape, the level of filling and the characteristics of the tank motion, for example amplitude and frequency content, make up the principal parameters that determine the nature of the free surface flow. The relative importance of the different parameters depends on the characteristics of the flow, i.e. the response. There is a clear difference between sloshing in a shallow liquid condition and higher filling level conditions. For small ratios between liquid depth and tank length and an excitation frequency around resonance, a hydraulic jump or bore, which travels back and forth in the tank, is formed. When the steep front of the bore hits the tank wall, an impact occurs and a thin vertical jet shoots upwards. When the liquid depth is non-shallow and the liquid motion is two-dimensional, the free surface motion resembles a standing wave. Swirling or rotational flow is a special feature of three-dimensional flow, for instance, in a spherical tank, a vertical circular tank or a square-based tank.

When the interior tank surface is smooth and there are no internal structures as stringers obstructing the flow, the viscous damping of the resonant fluid motions is small as long as wave breaking does not occur. The damping for a smooth tank can be very small at finite depth, i.e. it takes a very long time for transience to die. The damping increases in general with decreasing liquid depth. Large amplification of the liquid motion occurs in a resonant condition. This leads to important nonlinear liquid behaviour inside the tank associated with nonlinear transfer of energy between different modes of liquid motion. Secondary resonances may matter. This means that higher harmonic oscillations caused by nonlinearities excite resonance periods lower than the primary resonance period(s). Secondary resonance increases in importance with decreasing liquid depth and increasing excitation amplitude.

It is popular to use Computational Fluid Dynamics (CFD) to model sloshing. We will address this approach in a more general way later in a separate section. However, the analytically based multimodal modal time domain method (Faltinsen & Timokha, 2001; Faltinsen et al., 2005) is more suitable to understand the many different flow configurations that can occur during sloshing. The assumptions are potential flow, no overturning waves and infinite tank roof height. The tank wall surface must be vertical in the free surface zone. The method is extremely fast from a computational point of view. The method has been derived for two-dimensional flow in rectangular tanks and for three-dimensional flow in prismatic tanks with a rectangular base. The effect of small chamfer at the tank bottom has been studied in a two-dimensional case. The method has been extensively validated by experimental results for wave elevation, lateral force and roll moment. The 2D studies have examined how the flow changes physical character by going from finite through intermediate to shallow liquid depth. The excitation amplitude is also an important parameter. The physical dissipation increases with decreasing depth.

The steady-state solutions may have several branches. Some of the branches may be unstable. We cannot have steady-state solutions along an unstable branch. Jumps can happen between different branches of solutions.

The multimodal method has been applied to three-dimensional flow with a square-based tank. There are three types of possible dominant steady-state wave response for longitudinal harmonic excitation, namely, 'planar' (two-dimensional), 'swirling' (rotary motions) and so-called 'squares'-like three-dimensional steady-state waves formed by a combination of diagonal wave patterns. Even with longitudinal excitation along a tank wall, 3D waves will occur due to non-linear interaction in a nearly square-based tank.

By adopting a stability analysis scheme, one can calculate effective frequency domains for the different wave behaviour and find critical depths where either the frequency domains of stable regimes or their wave response may change dramatically. Summarized theoretical and experimental results are presented in Figure 2 for longitudinal forcing along a tank wall. Theoretical and experimental effective frequency domains of different steady-state wave motions agree.
well. For fluid depth \( h = h_1 = 0.337l \) (\( l \) is the tank length), they have the same qualitative structure as for resonant sloshing in a circular basin, namely, the small neighbourhood of the primary resonance consists of two zones, where stable ‘swirling’ and irregular (no stable steady-state solutions) motions are realized. Stable ‘planar’ sloshing occurs for excitation frequencies slightly away from the main resonance.

Both prismatic and spherical tanks are commonly used on LNG carriers. The most important load on a spherical tank is the sloshing-induced hydrodynamic force, the predominant component of which is the lateral force. Swirling occurs readily so there is in general a force component perpendicular to the forced oscillation direction.

4. SLAMMING

Slamming is of concern in many marine applications. A special type of water impact is water entry of free-fall life boats. The deceleration, slamming loads and hydrodynamic loads on the top cover of the life boat are important response variables. This issue has lately got increased attention for life-boats on platforms in the North Sea. A too low maximum operating sea state can have serious economic consequences due to stop in oil and gas production.

Figure 2: Theoretical ranges of stable steady-state motions for longitudinal excitations of a square-base tank; depth/breadth ratio \( h/a \) versus excitation/lowest natural frequency \( \omega / \omega_1 \) with forcing amplitude/breadth ratio 0.0078. Shaded area = ‘chaos’ (vanishes for \( h/a < 0.25 \)). Experiments (\( h/a = 0.508 \) and 0.34): ‘o’-‘planar’ waves, ‘*’-‘swirling’, ‘#’-‘chaos’, ‘S’-‘squares’-like.

This zone falls into two non-connected regions, i.e. for lower and larger excitation frequencies than the primary one. The left region (for lower frequencies) contacts with the frequency domain of irregular waves (‘chaos’), while the right region partly overlaps the effective domain for ‘swirling’.

The effective domain for ‘squares’-like modes is in general overlapped with left region of ‘planar’ waves. Moreover, its steady-state amplitude is significantly larger than the amplitude of the ‘planar’ waves. Therefore, experiments will most probably not capture steady-state ‘squares’-like waves with a static initial liquid state. The situation changes with decreasing liquid depth-to-tank breadth ratio \( h/a \). The frequency domain of stable ‘squares’-like waves then occupies the small vicinity of the primary resonance instead of the regions of irregular waves, while the ‘swirling’ frequency domain moves away from the primary resonant zone. The frequency domain with three-dimensional flow increases with increasing forcing amplitude. Analysis as this will give important guidance for CFD simulations.

Steep waves impacting on a vertical tank wall represents an important consideration for shallow and intermediate liquid depths of a tank. An example is hydraulic jumps that can be formed at resonant conditions for shallow liquid conditions. It is common to analyze slamming in a tank by assuming a two-dimensional flow. However, swirling may cause important impact against the tank roof corners for non-small liquid depths in a nearly square-based tank.

Both local and global slamming effects must be considered. This is illustrated in the case of bow slamming on a ship in Figure 4 where the resulting global elastic vibrations (whipping) of the ship are exaggerated. The local slamming analysis for a ship is typically done by first calculating the ship motions without impact and then consider the impact with given conditions for the impacting hull. However, this procedure does not account for the mutual interaction between slamming and the global ship behaviour and needs to be improved in the future.

Many physical effects may have to be considered such as gas cushions, liquid compressibility and hydroelasticity. When analyzing slamming, one must always have the structural response in mind. An important consideration is the time scale of a particular hydrodynamic effect relative to wet natural periods for structural modes contributing significantly to large structural stresses. If the time scale of a hydrodynamic effect is very small relative to important structural natural
Figure 3. Tank roof impact with either high velocity jet or oscillating air cavity for a tank with large filling ratio.

periods, the hydrodynamic effect can be neglected. When the hydrodynamic loads occur on a time scale of important structural natural periods, hydroelasticity must be considered. This implies that the fluid (liquid, gas) flow must be solved simultaneously with the dynamic elastic structural reaction.

Figure 4: Artist's impression of bow slamming causing global elastic vibrations (whipping) of the ship hull.

Local external slamming effects on ship structures of steel and aluminum are first discussed. Drop tests with horizontal plates with correctly scaled elastic properties of steel and aluminum on calm water as well as varying wave conditions and impact conditions have shown recorded maximum pressures with very large variations for a given drop speed. A maximum pressure of about 80 bar was recorded with a drop speed of 6 m/s. However, the recorded strains were not sensitive at all and showed a time variations dominated by the lowest beam mode. The maximum strain occurred approximately one quarter of the highest natural period after the impact. The effect of the impact was to cause a force impulse to the plate. The very high slamming pressures are unimportant, i.e. they cannot be used as a static loading and there is no correlation between pressures and strains. Numerical simulations of water entry of wedge-formed ship cross-sections with stiffened plating have shown the importance of the ratio between the loading (wetting) time and the highest natural period for the structure. If these results are translated into realistic ship structural dimensions and relative vertical motions between the ship and the waves, a rough guidance is that local hydroelasticity should be considered when the angle between the impacting free surface and the hull surface is less than 5 degrees.

We now turn to slamming in ship tanks. The discussion has also relevance for slamming in other applications. It is common in tank design to do model tests for sloshing induced slamming effects by means of forced oscillation tests. However, the scaling of the model test results represents a challenge due to the many physical effects that may matter.

Because sloshing is associated with gravity waves, we must require the Froude number is the same in model and full scale. Further, the wave induced ship motions that excite sloshing, is also Froude scaled. If harmonically forced oscillation of the tank with frequency $\omega$ is considered, Froude scaling implies that $\omega \sqrt{L/g}$ must be the same in model and full scale. Here $L$ is a characteristic tank dimension such as the tank breadth. A conventional model test approach does only consider the effect of Froude scaling. However, other scaling parameters of possible importance are summarized below.

If hydroelasticity matters during impact, we must ensure that the relevant natural frequencies for the elastic structural vibrations are Froude scaled. For instance, let us consider a steel tank. The bending stiffness matters for the natural elastic frequencies of importance. Further, the length of the elastic plate must be geometrically similar in model and full scale. The scaling of the natural frequency may be achieved by having different bending stiffness $EI$ in model and full scale. The bending stiffness may be properly scaled by considering a different material and/or changing properly the thickness of the material. Because the main interest is to find the slamming induced structural stresses, care must also be shown in scaling structural stresses. More structural modes may be needed for membrane structures than for flexible structures. Some of the important structural modes for membrane structures may have relatively lower natural periods than for flexible structures.

Figure 5 shows a steel tank model in the MARINTEK sloshing test rig. The model is relatively narrow to achieve a two-dimensional flow. Stiffened steel plates
ensure rigidity of the tank. The tank is forced to oscillate harmonically in the horizontal plane with a period close to the highest natural period for the fluid motion. The aluminium plate is clamped in the upper and lower part, and is free to move along its vertical sides. The natural frequencies for the aluminium plate have been Froude scaled. The full scale lowest natural frequency is comparable to that of a similar structural element in a typical OBO-carrier tank. A mathematical model based on beam theory and Wagner’s impact theory has been developed which account for the hydroelastic behaviour. The measured structural damping is input to the mathematical model. Good agreement with the experiments is documented in Figure 6.

If slamming is associated with the formation of gas pockets, the Euler number must be the same in model and full scale. The Euler number is defined as $Eu = p_\ast / \rho U^2$. Here $U$ is a characteristic velocity and $p_\ast$ is the ullage pressure, i.e., pressure in the ambient gas of the tank. The consequence of both Froude and Euler number scaling is that the ratio between $p_\ast$ in model and full scale is equal to the ratio between the length scale $L$ in model and full scale.

We will give a more detailed discussion of impact with air cushion and consider a scenario as in Figure 3. Two-dimensional flow and a flat rigid surface are assumed. The entrapped gas occurs due to the geometry of the impacting free surface and not due to the effect of the gas flow on the liquid flow. A gas pocket has a natural frequency due to the compressibility of the gas and a generalized added mass associated with the liquid oscillations caused by the gas cavity oscillations. Resonant oscillations of the gas cushion are excited...
during the impact. The consequence of the oscillations of the gas pocket is that the liquid pressure will oscillate with the same frequency. This is illustrated in Figure 7 in terms of slamming pressure measurements at the tank roof. Several pressure measurements are shown indicating that the pressure is nearly constant within the air cushion. The natural frequency of the gas pocket oscillations will change with time. The power spectrum of the pressure recording shows that the energy of the pressures is concentrated around a narrow frequency range which is related to the time variation of the natural frequency. A particular concern is that the time dependent behavior of the gas cushion can cause hydroelastic effects. Acoustic wave radiation and thermal conductivity cannot explain the fact that the gas pressure oscillations in Figure 7 are highly damped. One possible reason is air leakage from the cavity.

We will now theoretically analyze the flow in the liquid caused by the impact. The corresponding velocity potential is denoted \( \varphi \). In addition, we have the incident flow velocity \( V = V_0 + V_t \) which is assumed to be known. Certain simplifications are made in the analysis of the flow caused by the impact. The tank bottom and the tank wall opposite to the impact area are assumed far away relative to the length of the impact area. The consequence is that the tank bottom and the opposite tank wall are assumed at infinity in the analysis.

**Figure 7. Slamming pressure measurement when an air pocket is generated at the tank roof due to horizontal regular oscillatory tank excitation in side a rectangular tank with dimensions 0.6m × 0.6m × 0.1m.** The figure shows also the power spectrum of the pressure recording.

**Figure 8. Boundary value problem for the impact with gas cavity**
The boundary condition at the tank wall adjacent to the impact area is accounted for by mirroring the flow about the tank wall. The boundary value problem for the liquid flow caused by the impact is illustrated in Figure 8. A local coordinate system \((x, y)\) is introduced with origin at the corner between the tank wall and the tank roof in the impact region. The \(y\)-axis is vertically upwards. The wetted tank roof corresponds to \(y = 0\). The boundary conditions on the cavity surface and the free surface outside the impact are transferred to the \(x\)-axis. The dynamic free surface condition is \(\varphi = 0\) on \(y = 0\), \(x < -a\).

The liquid flow is modeled by a vortex distribution on \(y = 0\) between \(x = -a\) and \(x = a\). The vertical velocity acting on the gas cavity is \(a\) priori unknown and is a consequence of solving the problem. In addition, we use the nonlinear equations for conservation of mass in the gas based on an adiabatic pressure-density relationship and no leakage. The pressure in the gas cavity is equal to the hydrodynamic pressure at the interface.

\[
\frac{\sigma_n \sqrt{\Omega_0 \rho_l}}{\lambda p_u} = \begin{cases} 
0 & \text{for } b/a < 0.6 \\
1 & \text{for } b/a > 0.6 \\
\frac{b}{a} & \text{for } 0.6 < b/a < 1
\end{cases}
\]

**Figure 9. Non-dimensional natural frequency \(\sigma_n \sqrt{\Omega_0 \rho_l} / (\lambda p_u)\) for a gas cavity as a function of the length ratio \(b/a\) defined in Figure 8.**

In Figure 9 we have presented the non-dimensional linear natural frequency \(\sigma_n \sqrt{\Omega_0 \rho_l} / (\lambda p_u)\) as a function of \(b/a\). Here \(\Omega_0\), \(\rho_l\), \(\lambda\), \(p_u\) are the mean gas volume, mass density of the liquid, ratio of specific heat and ambient gas pressure, respectively.

We will use the experimental conditions for the experimental results presented in Figure 7 as a basis for further discussions. There are uncertainties related to the measurements of inflow conditions. The average upwards velocity of the free surface along the length of the air cushion is estimated to be 0.6 m/s at the time of impact, while the acceleration \((V_i)\) is \(-6.51\) m/s\(^2\). The 2D air cushion volume \(\Omega_0\) starts out at \(4.32 \cdot 10^{-4}\) m\(^3\). In addition, we set \(b = 0.0695 m\) and \(a = 0.07 + 0.781t < m\). The atmospheric pressure and the mass density of the liquid are \(p_t = 10^5 Pa\), \(\rho_l = 10^3 kgm^{-3}\), respectively.

We will elaborate on the fact that it is normal in tank testing to only Froude scale. This means that \(p_t\) is the same in model and full scale. We will illustrate the consequence of that in terms of scaling the pressure by using the described mathematical model.

\[
\begin{align*}
\frac{p_f}{p_m} & = \lambda \bigg( \frac{L_f}{L_m} \bigg) \bigg( \frac{\rho_f}{\rho_m} \bigg) \bigg( \frac{V_f}{V_m} \bigg) \\
\frac{\rho_f}{\rho_m} & = \bigg( \frac{\lambda}{\rho_f} \bigg) \bigg( \frac{\rho_m}{\rho_f} \bigg) \\
\frac{V_f}{V_m} & = \bigg( \frac{\lambda}{\rho_f} \bigg) \bigg( \frac{\rho_m}{\rho_f} \bigg) \\
\frac{p_f}{p_m} & = \lambda \bigg( \frac{L_f}{L_m} \bigg)^{0.7}
\end{align*}
\]

**Figure 10. Calculated ratio \(p_f/p_m\) between maximum total air cavity pressure in full scale and model scale presented as a function of the ratio \(L_f/L_m\) between full scale and model scale length.**

Figure 10 shows how the ratio \(p_f/p_m\) between maximum total air cavity pressure in full scale and model scale behaves as a function of the ratio \(\lambda = L_f/L_m\) between full scale and model scale length. Froude scaling of the inflow conditions has been made. This means that velocities in full scale have been obtained by multiplying model scale velocities with \(\lambda\). Length scales in full scale have been obtained by multiplying model scale length with \(\lambda\). Accelerations are the same in model and full scale. If the pressure should Froude scale, then the dynamic pressure in full scale should be obtained by multiplying the model scale pressure with \(\lambda\). Figure 10 shows that this provides a conservative estimate. Further, a linear analysis showed that the dynamic pressure should scale with \(\lambda\). This gives non-conservative results.

Nonlinearities play a more significant role in full scale than in model scale. This will be illustrated by choosing \(\lambda\) equal to 81. The previous model test conditions are mainly used as a basis However, the deceleration term \(V_i\) in the inflow velocity has been changed to \(-1.35 m/s^2\) to avoid negative inflow velocity during the selected full-scale simulation time of 4s. The results for the total pressure by the linear and nonlinear analysis are presented in Figure 11. The nonlinear model and the
linear model give very different time histories for full scale conditions while this is not true for model scale conditions. We note that the linear model predicts unphysical negative pressures during certain intervals. Further, the nonlinear model gives significant reduction in the pressure amplitude with time even though damping of the gas cushion oscillations has been excluded. The maximum predicted pressure by the nonlinear model is about 8 bar. The pressure changes rapidly with time in the initial phase. To what extent hydroelasticity matters depend on the important structural natural periods for maximum strains. The presence of nonlinearities is less evident in the air cushion volume than in the pressure.

Figure 11. Total air cavity pressure in full scale conditions based on scaling the model scale conditions presented in Figure 7 with length ratio \( L_f / L_m \) equal to 81.

\[
\begin{align*}
    b &= 5.6295 \text{m}, \quad a = 5.67 + 7.029t \text{ (m)}, \\
    \Omega_0 &= 2.8344 \text{m}^2, \quad V_0 = 5.4 \text{ms}^{-2}, \\
    V_i &= -1.35 \text{ms}^{-2}.
\end{align*}
\]

We will apply the results to the upper part of the transverse bulkhead in an OBO-carrier as illustrated in Figure 12. The tank roof is very stiff so that impact loads on the tank roof is of minor concern. A vulnerable area for the effect of tank roof impact is the vertical stiffeners at the tank wall adjacent to the impact. The structural response of an equivalent beam of uniform thickness with length 9.5m and with the upper end at the tank corner is studied. Steel is used as a basic material with Young modulus \( E = 210 \cdot 10^3 \text{Pa} \). The effective flange was chosen as the distance between the vertical stiffeners, i.e., 0.91 m.

It is well known from the analysis of a simple mass-spring system that the ratio between the duration of the loading and the natural period plays an important role in determining the dynamic amplification factor. There are many natural periods in our case, but we concentrate on the highest natural structural period 0.0749s. We should relate this period to the time history of the cavity pressure given in Figure 11. We cannot clearly define duration of the loading from this figure. However, the relatively rapid variation of the pressure during the initial phase should be of particular concern for dynamic amplification. Our calculations gave a dynamic amplification factor of 1.22 on maximum strains. However, if we properly change the natural period of the air cavity oscillations, a higher dynamic amplification factor can be obtained. We will present an example where the initial cavity volume was 0.75 times the original value. The other parameter remained the same. Figure 13 shows the resulting strains at the lower end of the beam both with a quasi-steady analysis and a hydroelastic analysis. Hydroelasticity causes clearly higher maximum strain. The dynamic amplification factor is 1.55. Because the large hydroelastic response is caused by the initial phase of the loading, the uncertainty related to the air cushion damping in Figure 7 is not a factor. The resulting stresses are far beyond yield strength even though we accounted for higher yield stress due to strain rate. It means that our linear elastic assumption is invalid. If the given impact scenario represents a design condition, we must obviously strengthen the tank wall in order to meet the criteria for allowable strengthen. This fact will also change the dynamic amplification factor. If the beam length is reduced to 3.5m and the other beam parameters are the same, the maximum stress is about 130 MPa with small hydroelastic effects. This is below an allowable stress of 175MPa for normal-strength steel.

However, we should not forget that our analysis assumes two-dimensional flow while violent sloshing in a tank can be strongly affected by 3D effects. Further, the analysis is based on scaling model test results with no internal structures. Internal structures as shown in Figure 12 will cause eddy making damping that lower the impact velocity on the tank roof and resulting stresses in

Figure 12. Details of structural arrangement in the tank of an OBO-carrier. Dimensions are in mm.
the tank wall. The pressure oscillations are nearly linearly dependent on the initial impact velocity.

Cavitation may also be of concern. The cavitation number is defined as \( \sigma = (p_c - p_a)/(0.5\rho U^2) \) where \( p_c \) is the vapor pressure and \( p_a \) is the ambient pressure at the cavitation position. We note a similarity between the cavitation number and the Euler number. Faltinsen (1997) reported on cavitation during drop tests of horizontal elastic plates on the free surface. The cavitation occurred because the vibrations of the plate caused negative pressures. However, the maximum stresses happened before cavitation started. The cavitation number in this case would be evaluated by setting the ambient pressure \( p_a \) equal to the atmospheric pressure. Boiling is the same as cavitation. Boiling takes place at the upper layer of LNG. Because LNG is boiling, the cavitation number must be considered in scaling model test results to full scale values. There is strongly limited knowledge about the effect of boiling on the slamming loads in an LNG tank.

The Cauchy number \( C_v = \rho U^2 / E_c \) characterizes the effect of compressibility on the flow. Here \( E_c \) is the bulk modulus for elasticity. The effect of liquid compressibility is less important in the case of a liquid with no gas bubbles and may be disregarded for a steel tank. This is a matter of what are the important wet structural natural periods. However, the speed of sound in a liquid can be substantially lower in the case of a mixture of gas and liquid. The consequence is an increased characteristic time scale for the effect of liquid compressibility. Because a mixture of gas and liquid can happen during violent sloshing and because LNG is boiling, we cannot rule out the effect of compressibility on slamming loads. The mixture of LNG and gas is not homogeneous. It is only the upper layer of LNG that is boiling. Because the sloshing and the impact affect the pressure distribution in the fluid and the pressure determines if cavitation occurs, the amount of bubbles is time dependent. The void fraction is therefore a function of time and space. This is an area requiring future research. Surface tension and viscosity are believed to be less important effects during slamming.

5. HIGH-SPEED MARINE VEHICLES

The focus is on the three main categories of high-speed marine vehicles supported by the submerged hull, foils or air cushions. It is important for high-speed marine vehicles to consider all hydrodynamic aspects in design. Considered items have to be limited in this presentation. A more comprehensive discussion of physical effects is given in the book by Faltinsen (2005).

An air cushion is enclosed between the two side hulls and by flexible rubber seals in the bow and aft end of a Surface Effect Ship (SES). The air cushion raises the vessel, depresses the water surface between the two hulls and lowers the zero-speed metacentric height. However, static stability does not represent a problem at zero speed. Further, the air cushion causes a wave resistance. Generally speaking, the resistance and needed power of a SES is clearly lower than a similarly sized catamaran at corresponding speeds. A danger is leakage from the air cushion in higher sea states with the consequence that the submergence of the vessel increases and the vessel speed drops. Other problem areas for an SES are wear of skirts as well as power peaks and wear and tear of propulsion/machinery systems caused by ventilation and cavitation due to the low submergence of the water jet inlet. The latter effect is most probable to happen in a seaway or during a turning manoeuvre. Wetdeck slamming and bow diving may occur. Further, berthing of an SES at high wind speeds may be difficult.

Cobblestone oscillations cause unpleasant vertical accelerations in small sea states and are the result of resonant compressible flow effects in the air cushion. Compressibility effects of the air in the cushion are essential. The oscillations are excited because the waves change the enclosed air cushion volume. It is the vertical vessel accelerations that are of concern. When cobblestone oscillations are not excited, i.e. for higher sea states, an SES will in general have good seakeeping behaviour relative to a similarly sized catamaran.

In reality one would use an automatic control system to damp out some of the "cobblestone" effect. This is done by controlling the air flow out from the cushion in such a way that it effectively acts as a damping on the system. In order to do that properly one needs a simplified but rational mathematical method that accounts for the dynamic pressure variations in the air cushion in combination with the global heave and pitch accelerations of the vessel.

Both the Euler and Froude numbers matter for cobblestone oscillations. A depressurized tank with a

![Figure 13. Elasic strain response at the lower end of an equivalent beam at the adjacent vertical wall of a tank roof impact with air cavity (Opedal, unpublished).](image)

Keynote 9 Faltinsen
wave maker generating high quality waves with small wave lengths is needed. Simplified numerical methods accounting for automatic control are commonly used instead of model tests.

A hydrofoil vessel with a fully-submerged foil system has in foilborne conditions in general good seakeeping characteristics, create small wash and have small speed loss due to incident waves. A hydrofoil vessel with a fully-submerged foil system needs automatic control to stabilize the vessel in heave, pitch and roll. Foils are normally designed for subcavitating conditions. However, the possibility of cavitation is then an important issue. Our discussion assumes subcavitating foils.

The Reynolds number scaling of foil tests and how to ensure turbulent boundary layers during model tests is an important issue. One effective way is to use Hama strips. The basis of a Hama strip is a tape. A saw-tooth shape is made on the upstream edge by means of a scissor.

Cavitation on foils designed for subcavitating conditions limits the vessel speed to the order of 50 knots. The pressure distribution along the foil should be relatively flat in order to minimize the possibility of cavitation, i.e. there must not be pronounced local pressure minima (suction peaks). The consequence of cavitation is that the material of the foil can be quickly destroyed and the lift capabilities of the foils can be significantly reduced. Another consequence is that the drag on the foils increases.

The importance of cavitation and ventilation for hydrofoil vessels will be illustrated by model test results for the side force coefficient of a strut used as a rudder on a hydrofoil catamaran. Ventilation means that air enters from the atmosphere to low pressure areas on the strut. It occurs typically at angles of attack \( \psi \) higher than 4° to 6° for a strut on a hydrofoil vessel in foilborne condition. However, this depends on the Froude number. Ventilation will cause a significant drop in the force. Hysteresis is associated with ventilation. This means that \( \psi \) may have to be lowered significantly to restore non-ventilated conditions. The detailed physical understanding of ventilation is limited.

Figure 14 shows model test results for a front strut-foil system used as a rudder on a hydrofoil catamaran. The model tests were done in the depressurized circulating free surface tank at the Technical University of Berlin. The side force coefficient \( C_L \) is presented as a function of the yaw angle \( \psi \) both at atmospheric air condition and at the cavitation number \( \sigma = 0.349 \) corresponding to the full-scale condition. The pitch angle is 2.7° and the submergence Froude number \( F_{hn} \) is 5.96 corresponding to a full-scale speed of 50 knots. The drop in the absolute

Figure 14. Model test results of the side force coefficient \( C_L \) of a front strut-foil system used as rudder on a hydrofoil catamaran (Minsaas, unpublished).
value of $C_z$ with either increasing positive $\psi$-values or decreasing negative $\psi$-values is an indication of the start of ventilation. This occurs, for instance, at $\psi \approx 4^\circ$ when $\sigma = 0.349$ and at $\psi = 6^\circ$ at atmospheric air condition.

A consequence of the ventilation at the strut is that air penetrates to the foil and causes a significant drop in the lift force on the foil. This drop occurred at $\psi$ approximately equal to $\pm 6^\circ$. The lift force on the ventilated foil was the order of 50% of the non-ventilated lift.

Increased heave and pitch damping of semi-displacement vessels is commonly achieved by using T-foils in the bow. Because ventilation and cavitation may limit the operability in higher sea states, wrong conclusions about seaworthiness can result from conventional towing tank tests. While the wave-induced response of displacement vessels can normally be adequately described by linear theory, this is not true for planing vessels. A linear 2.5D (2D+t) theory based on potential flow theory has become popular in describing wave resistance and wave induced motions and loads on semi-displacement vessels.

The steady wave pattern (wash) associated with fast semi-displacement vessels are of concern in coastal and inland waters. An example of the measured wave elevation in the near aft of a trimaran in deep water is shown in Figure 15. Free surface nonlinearities due to flow separation at the transom stern and breaking waves matter. Numerical methods have clear limitations in describing the wave field.

The water depth can have an important influence on the wave pattern when the water depth-to-ship length ratio $h/L$ is small. An important parameter is then the depth Froude number $Fh = U/\sqrt{\rho g}$ where $U$ is the ship speed. $Fh = 1$ corresponds to the critical speed. Large changes occur at the critical depth Froude number. The ship waves are very different for subcritical, critical and supercritical speeds in shallow water. The wave pattern at subcritical speed consists of transverse and divergent waves while only divergent waves exist for supercritical speed. The supercritical waves have a very small decay with distance which makes them of particular concern from a wash point of view. The angle (Kelvin angle) between the boundary of the wave system and the ship course is $19^{\circ}28'$ in deep water and when $Fh < 0.5 - 0.6$. A rapid increase in the Kelvin angle occurs for $Fh > 0.9$, and the angle is $90'$ for $Fh = 1$. False wall effects can therefore occur during shallow water model tests around the critical Froude number. If the vessel is in the vicinity of critical speed and in a channel, large waves can build up, be unsteady relative to the ship and travel faster than the ship. Russel described in 1865 that one day, the happiest of his life, something unexpected happened during his towing tests. A large solitary wave occurred ahead of the ship. He could not follow it by foot and got on the horseback and followed it for more than a mile.

Dynamic instability has increased importance with increasing Froude number. One reason is the increased importance of hydrodynamic pressure relative to hydrostatic pressure with increasing forward speed. There is a broad variety of dynamic stability problems. The loss of steady restoring moment in heel with forward speed can cause a sudden list of a round bilge monohull to one side. This can at high speed be followed by a violent yaw to one side. The consequence can be capsizing. This “calm water broaching” is the main reason round-bilge hulls should not operate beyond a Froude number of 1.2 (Lavis, 1980). The loss of steady restoring heel moment with speed should be accounted for in the design by having sufficiently high metacentric height at zero speed.

When a high-speed catamaran in following waves has a speed close to the phase speed of the incident waves,

![Figure 15. Model tests on high-speed trimaran in calm water. Wave elevation measurements at Froude number 0.4 and colour code for the ratio $\eta/L$ between the wave elevation and the ship length (INSEAN).](image)
the catamaran can come in a position relative to the waves so that the fore part of the vessel dives into a wave crest. If there is not sufficient buoyancy in the fore part of the vessel, a critical situation may occur.

Examples of dynamic instabilities for planing vessels are “corkscrew” pitch-yaw-roll oscillations, chine walking and porpoising. Porpoising is dynamic instabilities in the vertical plane and is commonly seen. Cavitation and ventilation in the aft body can cause dynamic instabilities.

The Froude number dependence and roll, sinkage and trim matter for manoeuvring of semi-displacement vessels. Strong nonlinearities may occur for a trimaran. For instance, steady heel may cause sign change in steady yaw moment for relatively small heel angles. Numerical methods have difficulties in describing those effects.

Global wave loads matter in the design of vessels longer than 50m. Important global loads for catamarans are transverse vertical bending moment (often called split moment), vertical shear force and pitch connecting moment, as illustrated in Figure 16. Torsional moments, vertical shear forces and vertical bending moments at transverse cross-sections are also of concern, as for monohulls. Both continuous wave loading and slamming will cause global loads. When wetdeck slamming-induced global loads are analyzed, the vessel must be considered elastic. The response is called whipping. When we consider the effect of continuous wave action, the catamaran is often considered as rigid in the determination of the global loads. However, it should be ensured in a linear analysis that the natural frequencies of the global elastic modes are higher than the encounter frequencies of practical interest. When the global elastic modes are accounted for during continuous wave loading, the phenomenon is called springing. Nonlinear effects may also cause springing.

![Figure 16. Examples on global wave loads on a catamaran.](image)

The largest global wave loads in a longitudinal cut of a catamaran occur in oblique sea conditions. The time window for testing in existing model tank facilities in oblique sea can be limited due to the high speed and the fact that transients must die out. Experimental error sources associated with non-constant wave conditions in a basin should also be considered.

### 6. COMPUTATIONAL FLUID DYNAMICS

It has become popular to use Computational Fluid Dynamics (CFD) to solve fluid flow. There is a broad variety of numerical methods. An overview is given in Figure 17. By CFD is meant that the fundamental governing equations, with initial conditions and

![Figure 17. Overview of numerical methods in fluid dynamics with emphasis on possible solution strategies within Navier-Stokes solvers (Greco, unpublished).](image)
nonlinear boundary conditions based on either potential flow, inviscid or viscid fluids are solved. For instance, we do not mean linear strip theories for ship motions.

We may ask: What are then the advantages and disadvantages of using CFD? Advantages are that complex structures and general excitation may in principle be considered. A CFD method may provide good flow visualization including details such as the vorticity distribution. Flow separation around structures can be simulated. It seems generally accepted that CFD codes have difficulties in predicting impact loads when the angle between the impacting free surface and the body surface is small. Actually hydroelasticity may then play an important role. Most computations do not consider this fact and assume a rigid structure.

Another disadvantage is that the CFD methods are time consuming which makes statistical estimates of response variables in a stochastic sea difficult. Some methods may not be robust enough. For instance, a Boundary Element Method breaks down when an overturning wave hits the underlaying free surface. Numerical problems may also arise with a BEM at the intersection between the free surface and the body boundary. When a BEM works, it is in general a fast and accurate method relative to other CFD methods. Nonlinear effects associated with water entry of two-dimensional bodies can in many cases be adequately described by potential flow theory and the Boundary Element Method.

CFD methods based on the Navier-Stokes equations are often very robust. Movies based on CFD may look convincing without always representing the reality. Care must be shown that the solutions are true physical solutions. If sufficient care is not shown, some of the methods may numerically lose or generate liquid mass on a long time scale. This is of particular concern in sloshing simulations. Because the highest natural period of the liquid motion is strongly dependent on the liquid mass, this can result in an unphysical numerical simulation. Verification and validation are a must. By verification is meant that the solutions are consistent with the governing equations, initial and boundary conditions that have been used. By validation is meant comparisons with model tests and full scale trials. Experiments are, of course, not free of errors. So it is important that the experiments are accompanied by an error analysis. Verification can be done by temporal and spatial convergence tests and by controlling that conservation of global mass, momentum and energy are satisfied. Comparisons should be made with analytical methods such as the linear potential flow theory.

Because commercial CFD codes are generic of nature and special physical features are specific for the different application fields, one cannot necessarily trust documentation of verification and validation in other applications than those one is interested in.

CFD methods may provide important insight into flow particulars. We will illustrate this by an example, where the Smoothed Particle Hydrodynamics (SPH) has been used in two-dimensional flow conditions. This story illustrates the importance of mutual interaction between analysis, CFD and experiments. However, let us first give some background for the story. The author was consulted in the development of a SPH code at INSEAN. He then wanted the code developer to control his results against the multimodal method and existing experiments. He, for instance, wanted to see how well jump phenomena in the steady state solution were predicted. The SPH code did well. However, there were some results for excitation periods from about $0.8T_1$ to $0.95T_1$ that did not agree well neither with the multimodal method nor the experiments. Here $T_1$ means the highest natural period. After lengthy check of the code without finding errors it was decided to do new experiments at INSEAN and those experiments did agree with the SPH simulations.

![Figure 18. Experiments (left photo) and SPH simulations (right) of a 2D rectangular tank near critical depth and for excitations near the lowest natural frequency. Colour codes represent the vorticity (Colagrossi, private communication).](image)
The tank was forced to oscillate horizontally with period $T^*$ close to the highest natural period $T_1$. The objective was to study steady state conditions. The fluid depth-to-tank length ratio $h/l$ was 0.35 which is close to the critical depth $h/l = 0.3368...$, where a change in flow conditions occur due to nonlinear effects.

Figure 18 shows both experimental and SPH results illustrating very steep overturning waves. The model tests also show strong mixing between air and liquid. Because the SPH assumed one-phase flow, the mixing of air and liquid was not predicted. The numerical simulations demonstrated a following decrease in the wave elevation with a later increase of free surface level to what is shown in Figure 18. This happens over a longer time than the forced oscillation period. The consequence was the presence of subharmonic terms in the lateral force during steady-state conditions. The multi-modal method cannot predict subharmonic behaviour. However, the subharmonic behavior was experimentally confirmed. The physical reason to the subharmonic terms was generation of vorticity. The SPH was used to explain this fact.

If a step should be made in improving seakeeping calculations, we have to rely on Computational Fluid Dynamics (CFD) and solve the Navier-Stokes equations. A graph was made by the author in 1988. Major previous achievements in computations of wave induced motions and loads were listed as a function of available computer power and year. A prognosis of the computer power in year 2000 was made and it was asked if it would be common to solve the 3D Navier Stokes equations in waves for ships and ocean structures. Even though the available computer power was underestimated, we are not at the stage where solving the Navier Stokes equations in waves is standard practice.

A domain decomposition method is suggested for strongly nonlinear wave induced flow at the ship. The violent flow near the ship including possible green water on the deck and water impact is handled by a CFD solver based on Navier-Stokes equation with two-phase flow while the more moderate free surface motion at the rest of the sea is solved by a Boundary Element Method (Figure 19).

The following example assumes the near-field solver to be a Finite Difference Method combined with a Level Set-method to describe the interface between water and air. Computational time is roughly estimated by Greco (unpublished) to be 16 hours for one wave period with a PC Pentium IV 2.8 GHz. If 1080 oscillation periods is assumed in a 3 hour storm, this means a computational time of approximately two years. Further, many realizations of each sea state as well as many sea conditions are needed in the design analysis. Computational speed may not be an obstacle with the combined use of supercomputers, future improvements in numerical methods and alternative design procedures by selecting time windows with extreme conditions. However, we have still not solved all physical problems needed for design such as ventilation, air cushions and proper turbulence modelling.

7. CONCLUSIONS

There is an increased focus on Computational Fluid Dynamics (CFD). A broad variety of numerical methods exist and it is difficult to favour one particular method to solve all marine hydrodynamic problems. A straightforward application of CFD in solving the behaviour of ship and ocean structures in severe weather is not conceivable in the near future if proper response statistics should be derived. CFD for high-speed vessels is, in general, not ready for design applications.

Simplified numerical methods based on rational physical approximations should be further developed. An example is on describing cobblestone oscillations of an SES.

A balance between analytical methods, CFD, model tests and sea tests is recommended. Further, links between the different disciplines of hydrodynamics as well as other topics such as structural mechanics and automatic control have been stressed.

Research on violent fluid motions such as slamming, green water on deck, sloshing in tanks, damaged ship with ingress/egress of water are important future activities. Knowledge about the effect of boiling on slamming loads in a LNG tank is needed. A physical understanding of what flow parameters matter for response variables of importance in design and operation is essential. The time scale of a particular physical effect in order for the structure to respond, for instance in terms of maximum stresses, is a basis for deciding what physical effects matter.

Dynamic instability, ventilation and cavitation are limiting factors for operation of high-speed vessels. There is limited knowledge about ventilation. A
depressurized towing tank with wave maker and 6DOF forced oscillator is desirable. Large horizontal tank dimensions may be needed for seakeeping and manoeuvring tests of high-speed marine vessels.

8. REFERENCES