25^{th} Symposium on Naval Hydrodynamics
St. John's, Newfoundland and Labrador, CANADA, 8-13 August 2004

Steady and Unsteady RANSE Simulations for Littoral Combat Ships

R. Azcueta (MTG Marinetechnik GmbH, Germany)

ABSTRACT

This paper presents steady and unsteady free-surface RANSE simulations for Littoral Combat Ships (LCS). The steady flow computations efficiently yield the complete resistance curve in one go – from zero to maximum boat speed – instead of computing only for one boat speed at a time. The dynamic sinkage and trim are also computed along with the resistance for the whole F_n-range. The unsteady simulations show that the method is robust enough and efficient for simulating large amplitude boat's responses to incident waves. This numerical study was carried out and presented to the client well in advance before model tests were performed. The posterior comparison with experimental data showed a surprisingly good agreement for the predicted resistance. Thus, this work represents a genuine performance prediction exercise. With CPU times of a few hours on common personal computers, this tool promises great potential for predicting the performance of high speed vessels like LCS.

INTRODUCTION

These days a widespread interest in the acquisition of LCS is shared by many navies worldwide. A main requirement for a LCS is to reach speeds in excess of 50 kn. To achieve these high speeds many advanced concepts are being considered, whereby the reduction of the wetted surface through planing hull forms or the use of lifting surfaces plays a significant role. Model tests are an obvious way of obtaining basic knowledge on the performance of such advanced concepts. However, numerical methods can prove to be useful tools for this task as well. Potential theory codes usually have accuracy limitations for high Froude numbers and for strongly non-linear flows. RANSE computations on the other hand are more accurate and have no restrictions for high speeds but usually do not take into account dynamic lift which is important for high speed craft and are not efficient enough in terms of computational time.

Therefore, having a numerical method capable of overcoming these two shortcomings would be highly desirable and could be of much use for concept design. This work presents computations with a RANSE-based method that addresses these two problems. It has shown to be very robust and efficient and it allows to obtain both dynamic sinkage and trim as well as the vessel motions in 6 degrees-of-freedom. The method has been applied to several dynamic cases showing that large amplitude motions and even ship capsizing can be simulated. Slamming cases, water entry, wave-piercing, water on deck and planing craft jumping in waves have been simulated and the results have been, whenever possible, successfully validated.

The robustness and efficiency of this method is mainly due to the simplicity of tracking the vessel's motions without deforming the numerical mesh or using complicated multimesh strategies. The Volume Of Fluid (VOF) method in conjunction with a moving, rigid mesh attached to the vessel and suitable boundary conditions are shown to be a good choice for these kind of applications. There are no restrictions on the vessel's speed and on hull shapes. Any hull configuration, like multihulls or hulls including lifting surfaces could be analysed.

MTG Marinetechnik GmbH initiated a study on behalf of the German Ministry of Defence to investigate the hydrodynamic performance of advanced platform types for LCS and to access the suitability of numerical methods for performance prediction at the design stage. For this purpose model tests (resistance and seakeeping tests) for a conventional monohull and two advanced platform types – a wave-piercer catamaran and a pentamaran – were ordered at the Hamburg Ship Model Basin (HSVA) and at the Ship Model Basin Potsdam (SVA). The resistance tests with the conventional monohull were established as a benchmark to validate three numerical methods from different institutions. At MTG RANSE computations with the above mentioned method were performed for the monohull and the results were presented to the client many weeks before the model tests were carried out at HSVA. The agreement of the computed resistance curve with the experiments was surprisingly good and will be presented here. In addition, the comparison of the RANSE results with the results obtained at HSVA and SVA with the potential flow codes will be presented. Finally, simulations of the LCS motions in head waves for two LCS design candidates – the monohull and a wave-piercer catamaran – will be presented.

NUMERICAL METHOD

To couple the fluid flow and body motions I extended the Navier-Stokes solver COMET with a *body-motion module*. COMET is a commercial code developed in Germany by ICCM GmbH, now a member of the CD Adapco Group, the developers of the well-known multi-purpose STAR-CD code. COMET was one of the first Navier-Stokes codes to implement a modern free-surface feature specially tailored for computing problems related to naval hydrodynamics such as flows including wave-breaking, sprays and cavitation, as well as turbulent flows with well proven turbulence models.

The general idea for coupling the fluid flow with the body motions is as follows: the Navier-Stokes flow solver computes the flow around the body in the usual way, taking into account the fluid viscosity, flow turbulence and deformation of the free surface. The forces and moments acting on the body are then calculated by integrating the normal (pressure) and tangential (friction) stresses over the body surface. Following this, the body-motion module solves the equations of motion of the rigid body in the 6 DOF using the forces and moments calculated by the flow solver as input data. The motion accelerations, velocities and displacements (translations and rotations) are obtained by integrating in time. The position of the body is updated and the fluid flow is computed again for the new position. By iterating this procedure over the time, the body trajectory is obtained.

Body Motion Module

Two orthogonal Cartesian reference systems (RS) are used: A non-rotating, non-accelerating Newtonian RS (O, X, Y, Z) which moves forward with the mean ship speed, and a body-fixed RS (G, x, y, z) with origin at G, the centre of mass of the body. The undisturbed free-surface plane always remains parallel to the XY plane of the Newtonian RS. The Z-axis points upwards. The x-axis of the body-fixed RS is directed in the main flow direction, i.e. from bow to stern, the y-axis is taken positive to starboard and the z-axis is positive upwards. The body motions are executed using a *single-grid strategy*, where a rigid, body-fixed grid moves relative to the Newtonian RS, and the ficticious flow forces due to the grid movement are automatically taken into account in the flow equations. The body-motion module is linked and run simultaneously with the flow solver and can operate and update all flow variables, boundary conditions and parameters of the numerical method.

The motions of the rigid body in the 6 DOF are determined by integrating the equations of variation of linear and angular momentum written in the form referring to G (all vector components expressed in the Newtonian RS):

$$m\ddot{\vec{X}}_G = \vec{F} \tag{1}$$

$$\overline{\overline{T}}\,\overline{\overline{I}}_G\overline{\overline{T}}^{-1}\dot{\vec{\Omega}} + \vec{\Omega} \times \overline{\overline{T}}\,\overline{\overline{I}}_G\overline{\overline{T}}^{-1}\vec{\Omega} = \vec{M}_G \tag{2}$$

where m is the body mass, $\ddot{\vec{X}}_G$ the absolute linear acceleration of G, $\vec{F}$ is the total force acting on the body, $\dot{\vec{\Omega}}$ and $\vec{\Omega}$ are the absolute angular acceleration and angular velocity, respectively, and $\vec{M}_G$ is the total moment with respect to G, $\overline{\overline{I}}_G$ is the tensor of inertia of the body about the axes of the body-fixed RS, $\overline{\overline{T}}$ is the transformation matrix from the body-fixed into the Newtonian RS.

The contributions to the total force and to the total moment acting on G are:

$$\vec{F} = \vec{F}_{flow} + \vec{W} + \vec{F}_{ext} \tag{3}$$

$$\vec{M}_G = \vec{M}_{G_{flow}} + (\vec{X}_{ext} - \vec{X}_G) \times \vec{F}_{ext} \tag{4}$$

where $\vec{F}_{flow}$ and $\vec{M}_{G_{flow}}$ are the total fluid flow force and moment determined by integrating the normal (pressure) and tangential (friction) stresses, obtained from the Navier-Stokes solver. They include the static and the dynamic components of the water and of the air flow. $\vec{W}$ is the body weight force. $\vec{F}_{ext}$ can be any external force acting on the body which one wants to introduce to simulate for instance the towing forces and moments.

The boat motions are described in each time instant by the position of its centre of gravity $\vec{X}_G$ and the body orientation given by $\overline{\overline{T}}$. Surge, sway and heave are defined in this work as the translations of G in the directions of the Newtonian RS. The angles of rotation are defined in the following order: First the rotation around the vertical axis in the Newtonian RS (yaw or leeway angle), second the rotation around the new transverse axis (pitch or trim angle), and last the rotation around the new longitudinal axis (roll or heel angle). To integrate in time the equations of motion a first-order explicit discretisation method has shown to work well and is used preferably. Instead of integrating the angular velocity $\vec{\Omega}$ to obtain the rotation angles, the new orientation of the body is found by integrating the unit vectors of the body-fixed RS, which are the columns of $\overline{\overline{T}}$. For details on the body-motion module see Azcueta (2001).

Flow Solver

The solution method in COMET is of finite-volume-type and uses control volumes (CVs) with an arbitrary number of faces (unstructured meshes). It allows cell-wise local mesh refinement, non-matching grid blocks, and moving grids with sliding interfaces. The integration in space is of second order, based on midpoint rule integration and linear interpolation. The method is fully implicit and uses quadratic interpolation in time through three time levels.

The deformation of the free surface is computed with an *interface-capturing scheme* of VOF type (Volume Of Fluid), which has proven to be well suited for flows involving breaking waves, sprays, hull shapes with flat stern overhangs and section flare, etc, (Azcueta, 1999). In this method, the solution domain covers both the water and air region around the hull and both fluids are considered as one effective fluid with variable properties. An additional transport equation for a void fraction of liquid is solved to determine the interface between the two fluids. The *High-Resolution-Interface-Capturing* (HRIC) discretisation scheme for convective fluxes in the void fraction equation is used to ensure the sharpness of the interface.

The solution method is of pressure-correction type and solves sequentially the linearised momentum equations, the continuity equation, the conservation equation of the void fraction, and the equations for the turbulence quantities. The linear equation systems are solved by conjugate gradient type solvers and the non-linearity of equations is accounted for by Picard iterations. The method is parallelised by domain decomposition in both space and time and is thus well suited for 3-D flow computation with free surfaces – especially when they are unsteady, as in the case of freely-floating bodies – since they require a lot of memory and computing time. For details on the flow solver see Peric (1996).

Previous Applications and Validation

This method has been extensively applied to naval hydrodynamic problems, some of which are:

- Resistance, sinkage and trim of Wigley and Series 60 hull models in strait ahead and drift condition
- Resistance comparison of two candidate designs for a navy support vessels
- Breaking waves around a fat ship model with a blunt bow similar to a tanker
- Drop tests with a wedge used for slamming investigations (Azcueta, 2001)
- Motions of the model of a naval combatant in head waves with an emphasis on slamming and water on deck
- Motions in waves of a dock-well navy vessel focusing on the sloshing in the dock-well
- Study on the efficiency of floating breakwaters interacting with waves in shallow water
- Two candidate designs of a mega yacht at full scale undergoing severe slamming in head waves
- Performance prediction at full scale of an IMS sailing yacht in calm water and in waves coming from any direction (Azcueta, 2002)
- Resistance prediction and motions in head waves (jumps) of a planing hull for speeds of up to $F_n = 4$ (Azcueta, 2003)
- Investigations of the dynamics of very large container vessels (up to 360 m length) sailing in extremely shallow water with 0.5 m under-keel clearance (results to be published)
- The main application area currently is the prediction of hydrodynamic performance of America's Cup and Volvo70 yachts.

LCS MONOHULL

Table 1 shows the main particulars of the monohull LCS at full scale and at model scale (1:24). A preliminary design for this monohull was completed at MTG including the hull lines, weight estimates, propulsive arrangement, system performance and costs. Figure 1 is a photo of the model used at HSVA for the resistance, propulsion and seakeeping tests.

Table 1: LCS main particulars

	full-scale	model-scale
Lenght water line L_{wl}	126.0 m	5.25 m
Breath B	16.10 m	0.67 m
Draft d	3.98 m	0.166 m
Mass m	3877 t	288 kg
KG	7.60 m	0.317 m
Pitch moment of inertia	$1.9 \cdot 10^6$ t m^2	240 kg m^2

Figure 1: LCS monohull model at HSVA

NUMERICAL MESH AND SIMULATION SET-UP

Two numerical meshes were generated using the ICEM-CFD Hexa mesh generator. One was optimized for the resistance prediction calculations with better resolution in the boundary layer and a total of 286 000 cells for one half of the vessel and a second one was optimized for simulating the vessel's motions in incident waves, with smaller expansion ratios and a total of 275 000 cells. The computational domain extends for about $1.4\,L_{wl}$ in front of the bow and behind the transom, $0.3\,L_{wl}$ above deck, $1.2\,L_{wl}$ below the keel and to the side. The mesh has such a large domain, especially above deck, in order to allow large pitch motions in head waves. Figure 2 shows a perspective view of the mesh used for the motion simulations.

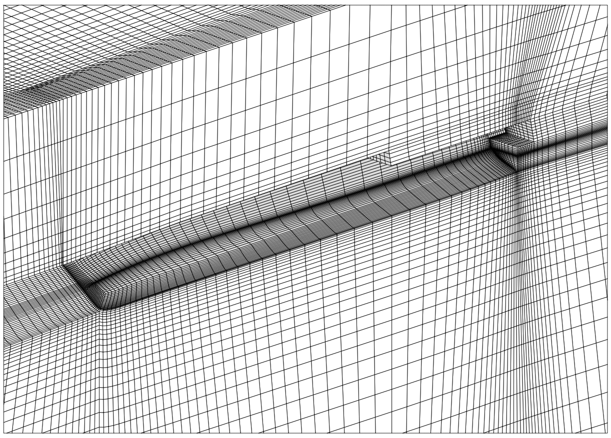


Figure 2: Numerical mesh for the LCS monohull

The pitch radius of gyration was set at $k_{yG} = 0.25\,L_{wl}$. The front, side, bottom and top flow-boundaries were specified as an inlet of constant known velocity (boat speed in opposite direction plus orbital velocity of the incident waves) and known void fraction distribution defining the water and air regions (wave elevation). The wake flow-boundary was specified as a zero-gradient boundary of known pressure distribution (hydrostatic pressure). All calculations were performed at model scale using the standard k-ϵ turbulence model with wall functions ($R_n < 3.7 \cdot 10^7$).

Both the resistance calculations and the simulations of motions in waves were carried out without having modelled the water jet tunnels and without appendages. The resistance tests in the towing tank were also performed with the bare hull and sealed water jet tunnels so that the comparison with the CFD results can be carried out without corrections. A water temperature of 17 degrees C was used in the computations. This temperature was estimated using the historical temperature data for the tank and resulted to be correct afterwards so that no corrections of computed results were required. The seakeeping tests were performed with the self-propelled model with model water jets.

RESISTANCE TESTS

RANSE computations are usually carried out for a given boat speed at a time and then repeated for as many speeds as are of interest. Here, a different approach is used: the entire resistance curve is computed in one single run. To achieve this, the boat, starting from the position at rest, accelerates very slowly until it reaches the maximum boat speed expected. Since the acceleration is small and the flow basically converges for each instant boat speed, the calculation can be considered to be quasi-steady. Note that although the flow is steady once converged, because the free surface has to develop its final wave pattern the computations (single-speed or accelerating) have to be carried out iterating in time, i.e. solving the transient terms of the flow equations.

Figure 3 shows the resistance test computed accelerating the boat from rest up to about 7.7 m/s (F_n up to 1.07). This corresponds to a full-scale speed in excess of 70 kn. However, there is no restriction in speed in this method and similar resistance tests have been carried out up to $F_n = 4$.

The red line in the figure represents the resistance curve. As mentioned earlier, a very important feature of these computations is that the dynamic sinkage and trim are computed throughout the entire F_n-range. These curves are given in Figure 2 as well (green and blue lines respectively). The red symbols also shown in Figure 3 are the measurement points from the experiments at HSVA. The experiments were carried out 3 weeks after the computed resistance curve had been presented to the client. The agreement is surprisingly good.

The CPU time needed for computing the resistance curve over the entire F_n-range is obviously greater than when computing only one boat speed, but it pays off if many boat speeds are to be computed. 28.5 hrs CPU on a linux-cluster using 4 AMD 2000+ processors were needed to simulate the resistance curve of Figure 3.

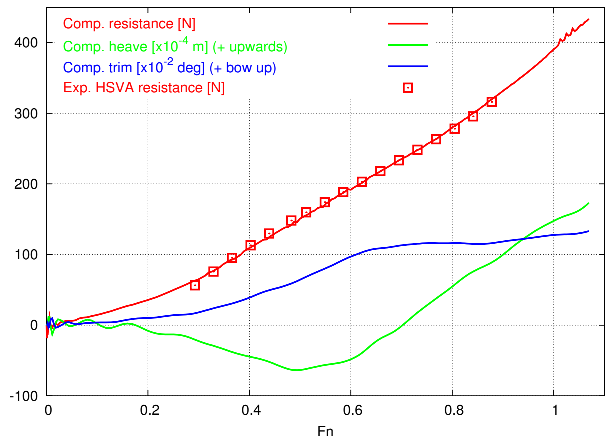


Figure 3: Numerical resistance test and computations at constant speeds

One important issue to take into account when performing these kind of resistance tests is to ensure that a constant *Courant Number* $c = v\Delta t/\Delta x$ is used for the entire F_n-

range. A value of $c < 0.5$ seems to be appropriate. The Courant Number is the ratio of the time step size Δt to the characteristic convection time, $v/\Delta x$, the time required for a disturbance to be convected a distance Δx. Since the mesh resolution giving Δx remains unchanged for the entire F_n-range and v is changing (the boat accelerates), Δt should be adjusted accordingly. This is achieved in these computations by setting $\Delta t = \Delta x_o / v$ or a minimum value for Δt when v tends to zero. Here Δx_o is a characteristic cell length, which is given as input at the beginning of the simulation.

The next issue to consider is that if a constant acceleration is used, in the high speed range where a small Δt is required, the boat's speed would change very slowly requiring too many time steps to reach the desired maximum speed. This is solved by gradually increasing the acceleration with increasing speed. The resulting curve shapes for Δt, acceleration and boat speed are given in Figure 4 as a function of the time step.

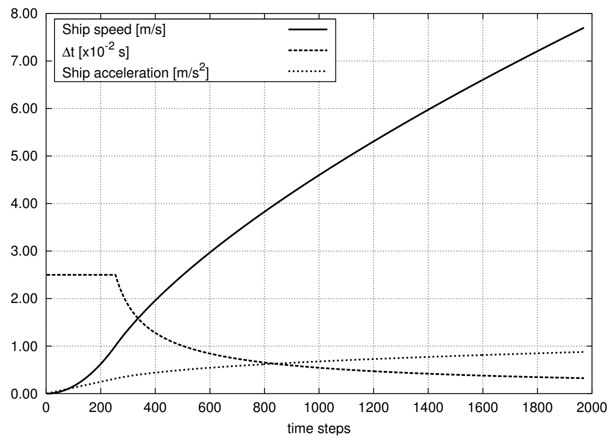


Figure 4: Curve shapes for boat speed, acceleration and Δt

In a previous study with the model of a planing hull the resistance curve obtained using this strategy was compared with computations for single speeds showing very good agreement. Furthermore, the resistance test was repeated with the boat decelerating from maximum to minimum speed using inverted function shapes for acceleration and Δt. Both resulting resistance curves were basically overlapping with the exception of a small F_n-range. This proved that the boat acceleration was small enough and that the additional forces due to the added mass were negligible (Azcueta, 2003).

Figure 5 shows a comparison of computed and measured resistance coefficients at model scale. In this figure it can be seen that not only the absolute total force in Newtons as measured and computed are in very good agreement as shown in Figure 3, but also the computed friction resistance coefficient ($C_F = R_{F_{comp.}}/(\rho/2v^2So)$ agrees very well with C_F from the ITTC'57 correlation line. Therefore, the computed pressure resistance coefficient and the residual resistance coefficient are in good agreement as well.

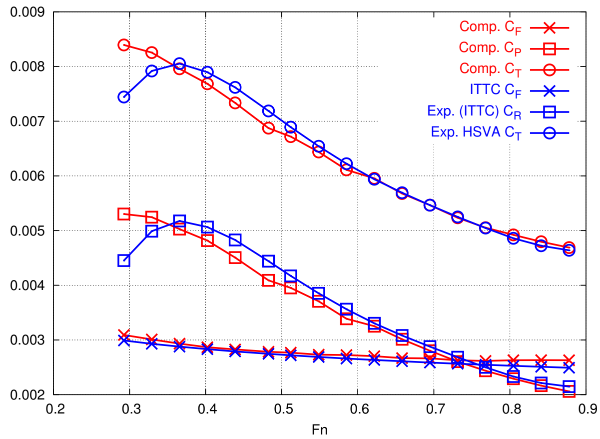


Figure 5: Comparison of computed and measured resistance coefficients at model scale

Finally, this LCS monohull design has been used as a benchmark for comparing the suitability of different numerical methods for predicting resistance. Figure 6 compares for the full scale the results of the RANSE computations with the results of two potential theory codes and the experiments at HSVA. The experimental residual resistance (Exp. (ITTC) R_R) was extrapolated to full scale using the ITTC'57 method with a form factor $k = 0$. The line labelled RANSE R_P is the computed pressure resistance extrapolated to full scale. The panel codes are "ν-shallo" by HSVA (line labelled Poti HSVA R_R in the figure) and "kelvin" by SVA (Poti SVA R_W).

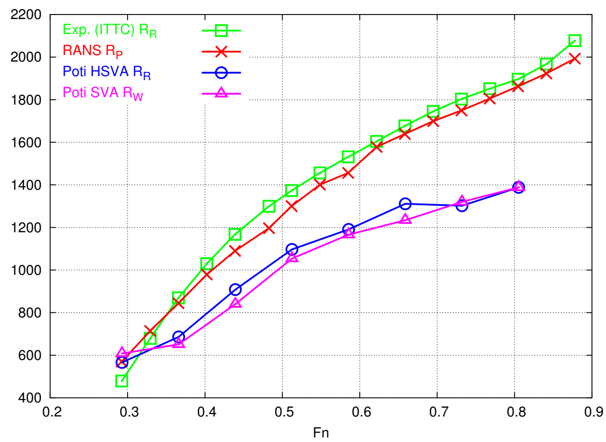


Figure 6: Comparison of RANSE pressure resistance with experimental and potential flow results

The panel codes underestimate the residual resistance by almost 30% for the higher speeds. Furthermore, the convergence of results for high speeds becomes increasingly difficult and is not possible beyond certain Froude number. On the other hand the RANSE computations show very good agreement throughout the entire Froude number range, which is most remarkable since the computation was (unlike the panel code ones) performed as a genuine prediction before the model tests were carried out.

LCS MONOHULL IN INCIDENT WAVES

The incident waves are generated at the inlet flow-boundary by imposing the instantaneous wave elevation and orbital velocities according to the linear wave theory. Three wave parameters are set at the beginning of a simulation: The wave amplitude ζ_w, the wave length λ_w and the wave direction μ relative to the boat course ($\mu = 0°$ means from astern and $\mu = 90°$ from port).

In the single-grid strategy used in these simulations, the computational domain moves as a whole relative to the undisturbed waterplane. The boundary conditions – the mean flow velocity, the orbital velocity, the void fraction distribution defining the wave elevations, the turbulence parameters and so on – have to be very carefully imposed at each time instant relative to the undisturbed waterplane. The VOF method and the implemented boundary conditions have proven to be very robust, since the free surface can leave the computational domain in any place, i.e. through the top flow-boundary in case that the boat heels or pitches with a large angle. Even the simulation of capsizing upside down is possible.

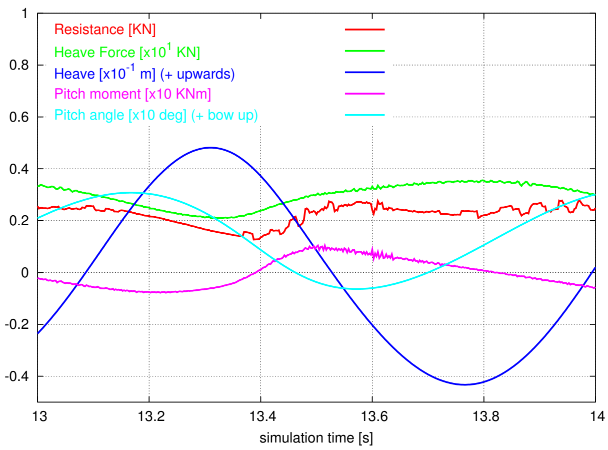


Figure 7: Forces and motions for $v_M = 3.94$ m/s, $\mu = 180°$, $H_w = 0.135$ m, $\lambda_w = 6.3$ m

Figure 7 shows motions and forces for one period of wave encounter for the model speed of 3.94 m/s in regular head waves of about 0.135 m height and 6.3 m length. Table 2 gives the corresponding values for the full scale. This wave height at full scale corresponds to the significant wave height for a sea state 5 in the North Atlantic. The wave length was chosen to be 1.2 times the length of the vessel, which comes close to the most unfavourable condition for pitch motions.

The diagram in Figure 7 clearly shows the very nonlinear nature of the motions. The diagram shows only a few characteristic results, but motion velocities, accelerations and local slamming pressures can be plotted as well. The averaged resistance for the one period shown in the diagram is 220 N and in calm water 169 N, i.e. the added resistance accounts for 30%.

At HSVA seakeeping tests for the monohull were performed for sea states ranging from 3 to 6 and speeds from 20 to 40 kn. Furthermore, a few runs in regular waves were performed in order to compare with the numerical predictions. The comparison is given in Table 2 as well. The pitch amplitude is in quite good agreement. The average trim and the heave amplitude are not so well predicted by the RANSE simulation (30% difference). One possible reason for the disagreement can be that in the seakeeping tests the model was self-propelled with model water jets and in the simulations the model was towed from the centre of gravity and it had no water jet tunnels, so that the centre of gravity is located further forward. The towing force in the simulations remained always horizontal, and the thrust in the physical model was parallel to the model longitudinal axis. This may influence the motions to some extent. Simulations with a simplified propulsive system should be performed for a more accurate validation.

Table 2: Comparison of motions for one wave characteristic

	computation		measurement
	model-scale	full-scale	full-scale
forward speed	3.94 m/s	37.5 kn	37.5 kn
wave height	0.135 m	3.25 m	3.25 m
wave length	6.3 m	151 m	151 m
encounter period	0.9 s	4.4 s	4.4 s
Pitch amplitude	1.86 deg	1.86 deg	1.95 deg
Trim average	1.27 deg	1.27 deg	1.70 deg
Heave amplitude	0.046 m	1.10 m	1.51 m

The CPU time required to compute one wave encounter period in this case is 2.5 hrs on 1 AMD 2000+ processor. At least 5 to 10 wave periods have to be computed to obtain periodic motions, which means that a simulation for a characteristic wave can be completed in one day on a common PC.

Figure 8 plots slamming pressures for two panels at the bow for one period of wave encounter. The pitch motion is also shown for reference. Figure 9 gives the position of the panels, which consist of several region elements at the wall. The pressure plotted in Figure 8 is an average of the pressures on the region elements. In this way, the average pressure is given for larger regions of interest for scantling purposes. The panel further up, labelled Nr. 2 in the figure, does not experience large slamming pressures since it remains almost always dry. The panel further down (Nr. 5) is more affected by slamming.

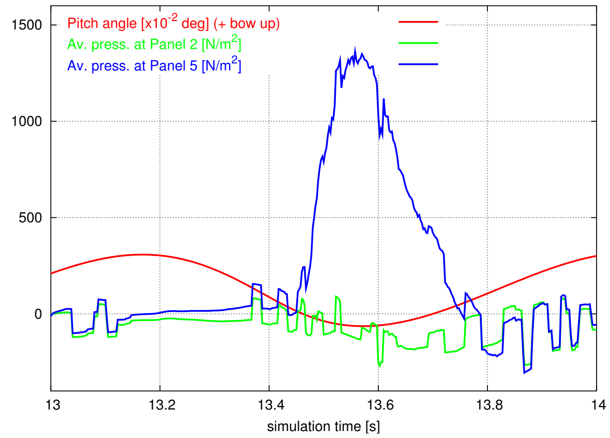


Figure 8: Pressure at selected panels

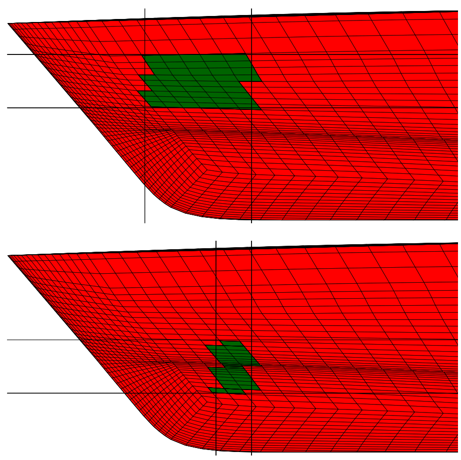


Figure 9: Position of the panels used for monitoring slamming pressures; top:panel 2, bottom: panel 5

Figure 10 shows a series of snapshots during one wave encounter period for the LCS monohull in head waves.

LCS WAVE-PIERCER CATAMARAN IN WAVES

Finally, simulations of motions in head waves were also performed for the wave-piercer catamaran which was one of the candidates investigated in the study on advanced LCS platforms concepts. The wave-piercer catamaran was designed to carry the same payload as the monohull and has therefore similar main characteristics as the monohull. A numerical mesh optimised for motions in waves consisting of 333 984 cells for one half of the vessel was also generated using the ICEM-CFD Hexa mesh generator.

One sailing condition in head waves was simulated with waves of 7.4 m height, 168 m length and a forward speed of 40 kn (full scale). The resulting motions for such extreme conditions were quite large: 1.6 m heave amplitude and 2.5 deg pitch amplitude with a period of 4.6 s. Figure 10 shows a series of snapshots of the simulation at model scale during one period of wave encounter. In the figures as well as in video animations wet deck slamming can be observed.

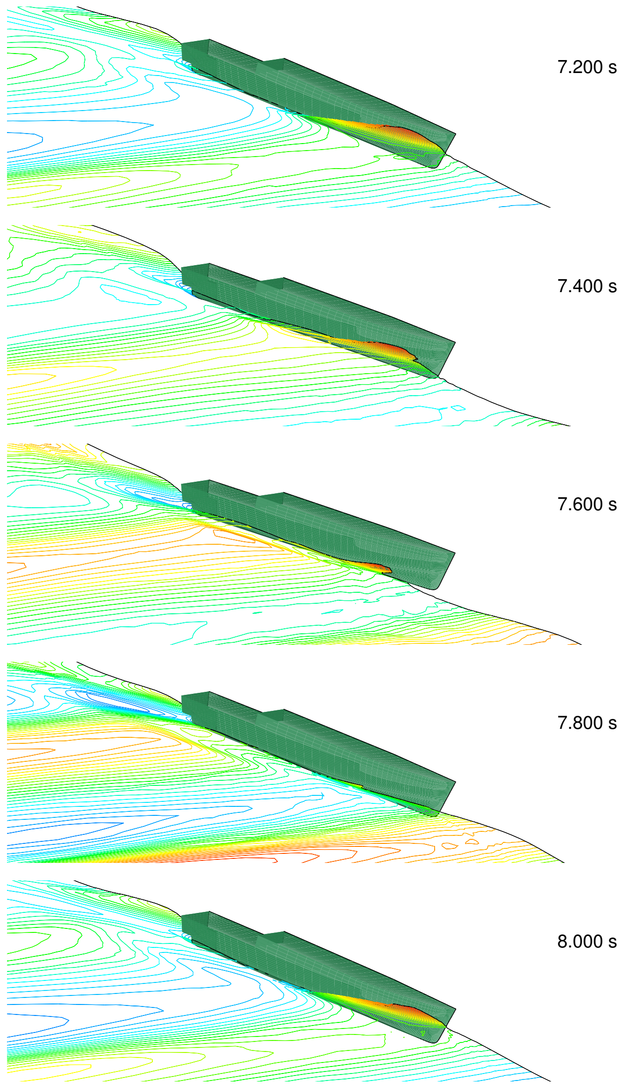


Figure 10: Snapshots during one wave period for the LCS monohull

ACKNOWLEDGEMENTS

The author would like to thank the German Ministry of Defence for allowing the publication of these results and the model basins HSVA and SVA for providing the results of the model tests and potential flow calculations. All rights for the result presented are reserved by the Federal Republic of Germany represented by the Bundesamt fuer Wehrtechnik und Beschaffung (German Ministry for Defence Technology and Procurement).

REFERENCES

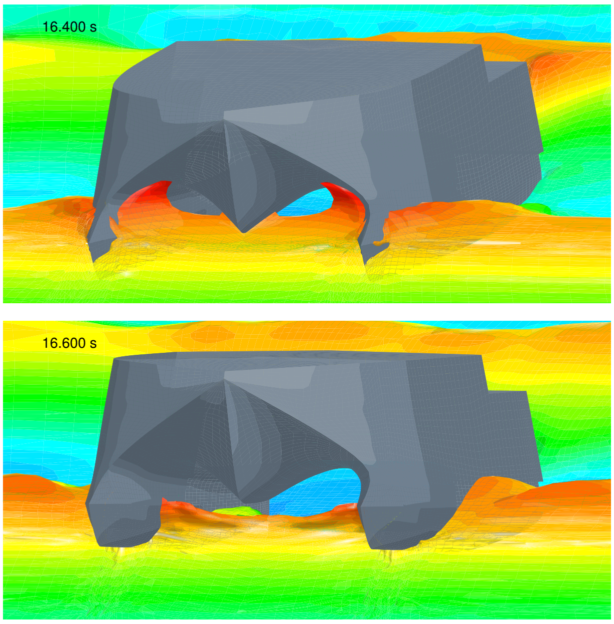
Azcueta, R., Muzaferija, S. & Perić, M., ”Computation of Breaking Bow Waves For A Very Fat Hull Ship”, *7th International Conference on Numerical Ship Hydrodynamics*, Nantes, 1999.

Azcueta, R., ”Computation of Turbulent Free-Surface Flows Around Ships and Floating Bodies”, PhD. thesis, Technical University Hamburg-Harburg, 2001.

Azcueta, R., ”RANSE simulations for sailing yachts including dynamic sinkage & trim and unsteady motions in waves”, *High Performance Yacht Design Conference* Auckland, 2002.

Azcueta, R., ”Steady and Unsteady RANSE Simulations for Planing Crafts”, *FAST Sea Transportation*, Ischia, Italy, 2003.

Ferziger, J. H. & Perić, M., ”Computational Methods for Fluid Dynamics”, Springer, Berlin, 1996.



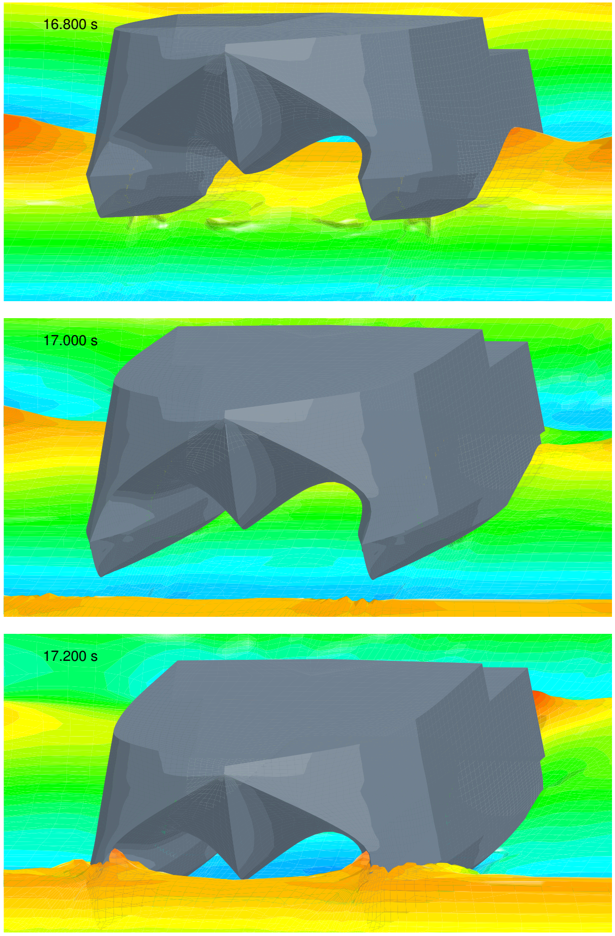


Figure 11: Snapshots during one wave period for the LCS wave-piercer catamaran in extreme conditions