Sea keeping simulation of fast hard chine vessels using RANSE

Mario Caponnetto
Rolla Research, Via Silva 5, Balerna (Switzerland)

Introduction

The design of an hard chine vessel requires, more then for other kind of boats and ships, a trade off between calm water resistance and sea keeping qualities. Following Savitsky theory the prismatic hull of minimal resistance, above a certain speed, if a flat hull. Even if we consider hulls more complex then a Savitky prism, the deadrise angle remains a fundamental parameter for calm water resistance and this angle should be minimized, whatever the other parameters are (beam, LCG…). Keeping all the rest fix, a flat hull generates more lift, allowing a lower trim angle (less wave drag) and less wetted surface (less friction drag). In waves a flat hull is one of the worst shape we can image. Large impact pressures are suddenly generated as the waves hit the bottom; this creates structural problems as well as vertical accelerations that can be hardly sustained by the crew. In practice a deadrise angle is always necessary but the amount of vee-shape depends on speed, boat dimensions and expected waves size. Modern boats are seldom prismatic, and have a deadrise angle that varies along the boat length. Depending on the applications, this angle may vary from 10/20 degrees at the transom to 20/50 degrees near to the bow, where impact pressure are expected to be higher. This so called "warped" hulls pose new problems to the naval architects. First, Savitky theory can't be used to calculate calm water trim and resistance of warped hulls. Second, there is a lack of data usable to predict sea-keeping qualities of warped (but even prismatic) hulls. Reasonably the more deadrise angle we give the better the sea keeping qualities, and worst the calm water performances, will be, but when to stop?
A couple of years ago we started trying to answer the problem of calm water calculation of fast hard chine vessels, bypassing the somehow crude approximations of the Savitky theory. The method we investigated to deal with this problem is a RANSE solver where the complex behavior of the free surface is computed with a special kind of VOF (Volume of Fluid) method. The commercial code ***Comet*** was tested among others, giving the best results (ref. 1, 2). Since then, many hulls have been tested, designed and optimized by our Company and the method is now a valuable tool for a number of leading shipyards in the nautical market. The step forward required by the designers is the possibility to compute sea keeping. The natural way to approach this problem is to use the unsteady and moving mesh capabilities of ***Comet***; some of the results obtained by now will be presented in the paper.

1. Overview of the method

1.1 Main features of the solver

For both steady (calm water) and unsteady (sea keeping) computations, a Finite Volume Method is used to solve the Navier-Stokes equations. Turbulence, of minor importance for this type of problem, is approximated using a K-epsilon model. The main feature of the code is the possibility to solve very complex behaviors of the free surface, typical of planing hulls. The Front Capturing Method built in the program can easily compute spray, breaking and overturning waves, detachment and reattachment of the flow along the chine, the side hull and the transom, and, in some extent, the ventilation of the hull. Details of the method can be found if ref. 3, 4.

1.2 Mesh generation

Structured or unstructured meshes can be handle by ***Comet***, as well as blocks of cells with not matching vertices. As far as the hull is a conventional hard chine vessel, we have found that the best results in terms of precision, as well as time spent to generate the mesh, are obtained with a structured mesh formed by hexahedral cells. This approach is somehow mandatory, as will be explained later, if the hull must be moved in a time domain simulation.

A Fortran code has been developed to build the structured mesh. The code loads the geometry of the hull and generates all the files needed to the solver (vertex coordinates, cells, regions…). The stretching of the cells in regions were a higher or lower resolution of the flow is required, can be easily controlled playing on a number of parameters. In general we want to increase cell density near to the hull and the free surface. Moreover the hull is placed in space giving in input a value for the trim and the sink, but in theory we can give all the 6 degrees of freedom. The hull, deck and transom are discretised. Special care is dedicated to the proper definition of the chine and the spray rails. Flaps, tunnels and skeg can be also modeled. A typical surface mesh of the hull is shown in figures 1 and 2.

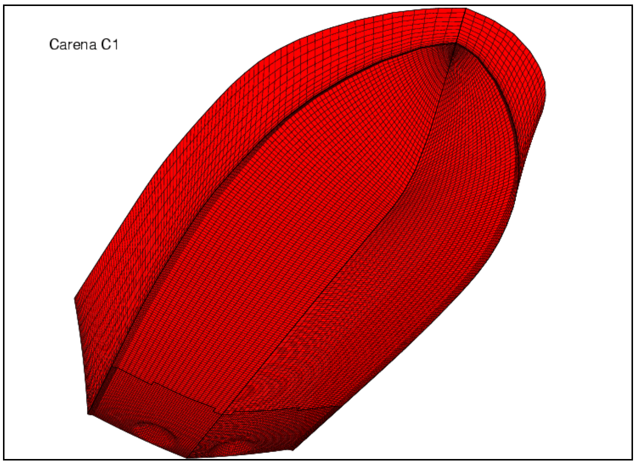


Figure 1

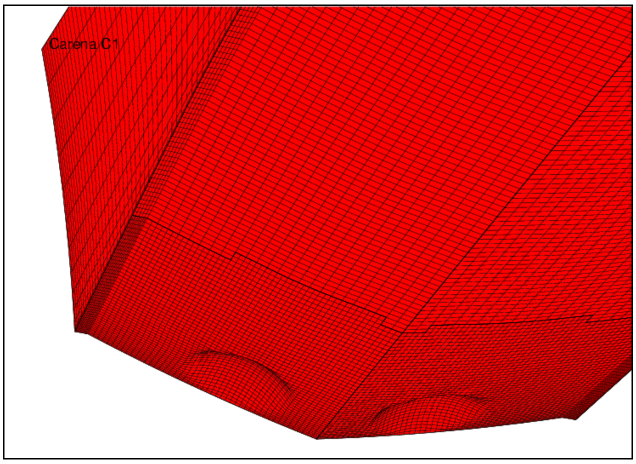


Figure 2

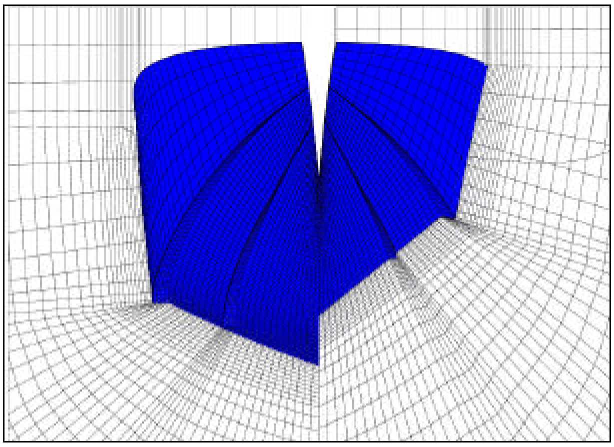


Figure 3

Figure 3 shows crosscuts of the volume mesh in two transversal planes near to the bow (right) and the stern (left). Local refinements can be done (as visible in figure 2) splitting a cell in a number of sub-cells.

The time required to build all the files needed to feed the solver is very short; about 10 seconds for a typical mesh having 200.000 cells.

1.3 Moveable mesh

In a sea keeping simulation the hull will change its relative position in the calculation domain at each new time step. By now we are dealing only with two degrees of freedom (trim and sink), since only bow waves are considered. Surge movements can be of some importance with higher waves, and can be added in the simulation, but this possibility haven’t been tested yet.

At each time step updated values for the trim and sink will be supplied to the mesh program to built the new mesh. The cell vertices along the external boundary of the computational domain remains fixed, while those over the hull are moved rigidly of the right amount. All the vertices in between will be consistently moved of a fraction of the hull movements, but keeping the same topology. Cells and vertices numeration remains the same. Each cell will keep its "identity" and will be formed by the same vertices, simply moved of a small quantity. In this way the solver can restart without the additional efforts needed to interpolate the old variable (pressure, velocity…) to the new cells; computational time and convergence rate is speeded up.

1.4 Boundary, initial conditions and time step

Regular bow waves or a "sequence" of waves with different lengths and highs can be generated. Since no surge movement is considered, the boat is fixed longitudinally in an "average" flow having a speed opposite to the boat speed Vb. Sinusoidal waves are enforced at the inlet; calling Lw the length of the wave and Hw its high, the wave profile at the inlet (h) will vary with time (t) in the following way:

$$h \,?\, \tfrac{1}{2} Hw * \cos(?_e t) \quad where: \;\; ?_e \,?\, ? \,?\, \frac{Vb}{g} ?^2 \quad and \quad ? \,?\, \sqrt{\frac{2?\,g}{L_w}}$$

At the inlet both air and water are blown in the domain. The cells whose centroid lies below h will inject water, the remaining air. Since h changes with time, there are cells that will change periodically the injected phase. The cells whose boundary surface is intersected by the local free surface will have a mixed phase proportional to the area below and above the free surface. Pressure and velocity boundary conditions are enforced. Above the free surface the air speed and pressure are set to Vb and Patm. Below the expressions for the pressure, horizontal and vertical velocities of a linear Airy wave (ref. 5) are used, but shifted from the undisturbed water level (h=0) to the actual free surface. This approximation has proven to produce in practice very regular waves, whose length and high can be easily controlled, as far as the slope of the wave is not too high.

The initial free surface is flat, the flow is initialized with V=Vb and the pressure is P=Patm for the air and P=Patm +rho*g*z for the water. This pressure condition is also applied to the outlet. To allow a smooth launch of the computation, the real wave high at in inlet is let to grow linearly from zero to Hw in about one encounter period.

Time step remains constant during the simulation and its value is normally set according to a Courant number of 0.15, calculated with the boat speed and the average length of the cells along the hull. For each time step a maximum of 30 iterations is allowed to reach convergence. The simulation is carried on until about 10 waves are encountered, but usually the phenomenon becomes periodic at the third or fourth encountered wave.

1.5 Forces and moments calculation

At the end of each time step the pressure over the hull is know and the vertical force and moment around the center of gravity can be calculated. Vertical and angular accelerations are then used to compute the new trim and sink for the next time step.

A problem arise from some numerical fluctuation of the calculated pressure distribution and forces between an iteration and the next one. Due to this small oscillations, even in calm water, a boat left free to heave and trim should manifest continuous small movement around the mean position. This problem has been found to be amplified in the unsteady calculations.

To prevent this phenomenon a quadratic least square fit of the vertical force (and moment) is performed on a number of time steps preceding the last one (figure 4). The fitting function is extrapolated to the next time step to compute an average force and moment suitable to calculate the trim and sink changes.

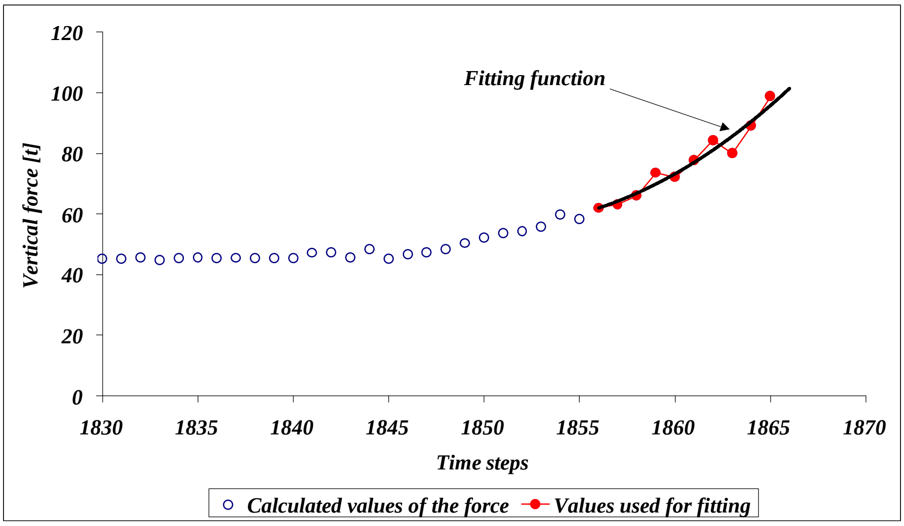


Figure 4

2. Sample cases

The code has been already used to perform sea keeping calculation for a number of planing hulls. Typically we have to deal with boats whose length ranges between 15 to 40 meters, and speed from 25 to 60 knots and above. For this kind of applications we have a lack of experimental data (in full or model scale) useful to validate the method.

What has been done by now is to check the robustness of the method and to perform convergence and mesh sensitivity tests. What can be said is that the accelerations computed are realistic. The code is very robust, since typically 5000 time steps are completed without crashes in the 90% of the cases. As far as accelerations are concerned, the results are converged with about 100.000 cells (for one half of the physical domain considering the lateral symmetry of the problem). Changes of hull geometry give qualitatively the expected results, and this is important in the perspective to rank different boats at the design stage.

It must be pointed out that, at difference of the typical behavior of conventional ships in waves, whose largest non linearity is the out coming of the bow off the water, very extreme situations have be encountered in our simulations. Sometimes the entire hull comes out of the water and then brutally hits the new incoming wave, rising the water above the deck.

Two cases are presented in the next figures as examples. In figure 5 a 24 meter boat moving at 45 knots is shown when coming near completely out of the water in a regular sea with Lw=30 m and Hw=2 m. Figure 6 shows the bow impact of a 35 meter yacht at 30 knots in similar wave conditions.

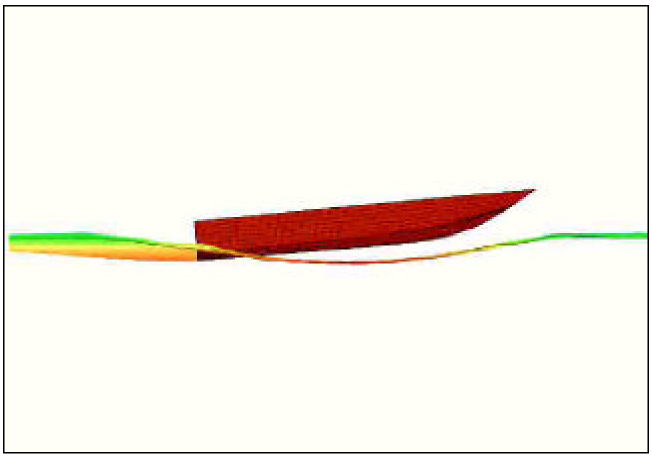


Figure 5

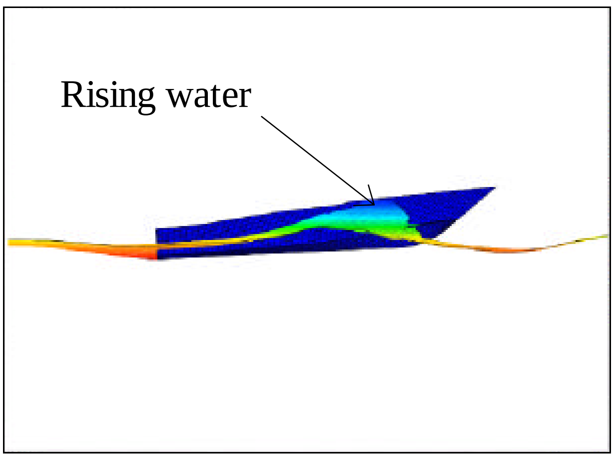


Figure 6

In these cases pressure impacts are huge, and probably to neglect hull flexibility, but instead considering the hull as a rigid body, doesn't respect sufficiently the real of the phenomenon. Moreover surge motion should probably be considered.

The next three figures show some samples of the trim, sink and vertical acceleration of the center of gravity time histories. The boat in question is 35 meter long and is running at 45 knots. Two waves are considered, Lw=20 m, Hw=1 m and Lw=30 m, Hw=1.5 m.

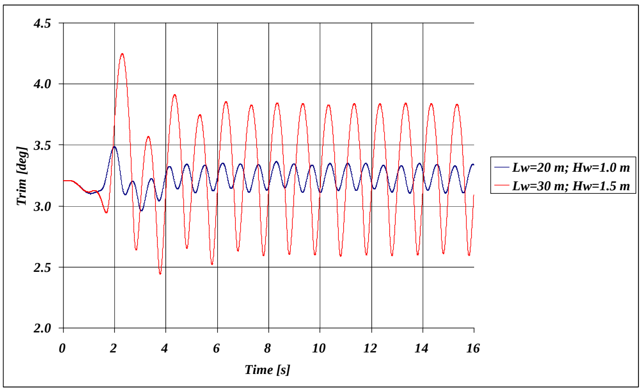


Figure 7

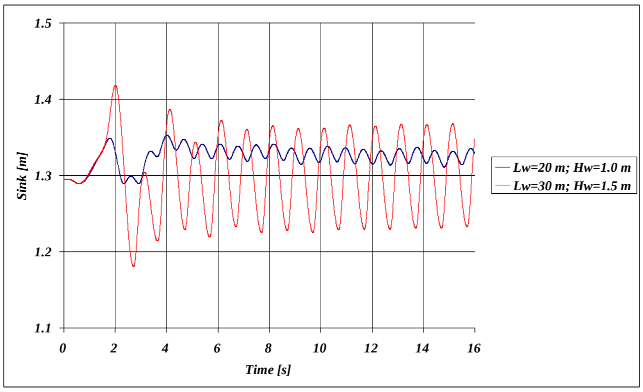


Figure 8

An other useful quantity derived is the time history of the drag, that integrated over time can give the "average" increase of drag, allowing an estimation of the speed reduction in waves. A sample is presented in figure 10, where the computation has been performed for a 90' Open yacht.

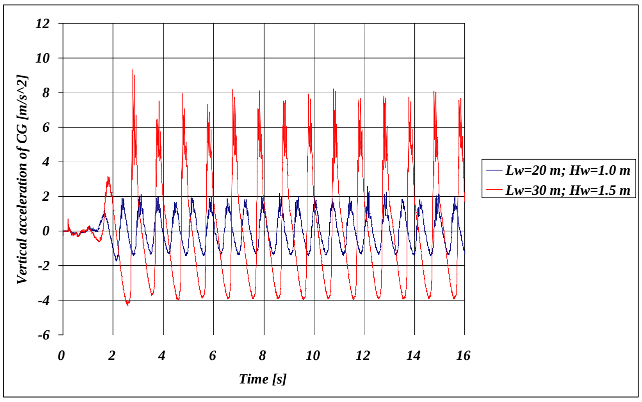


Figure 9

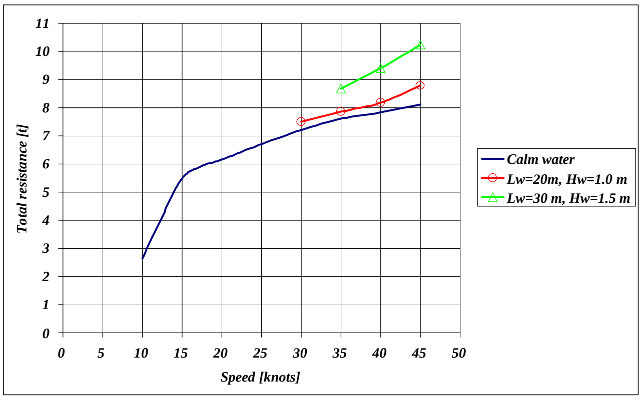
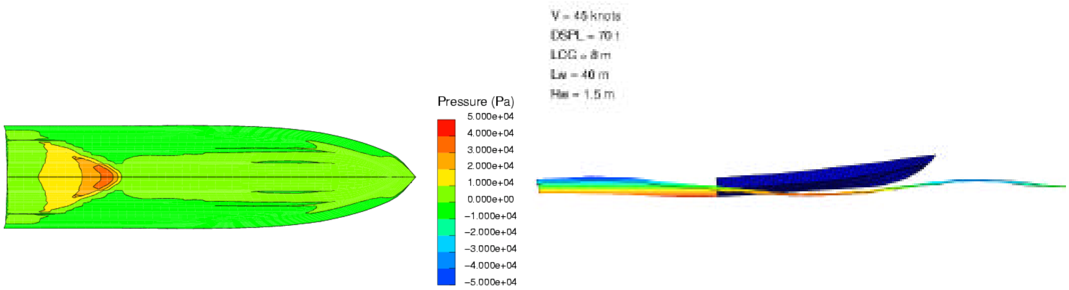
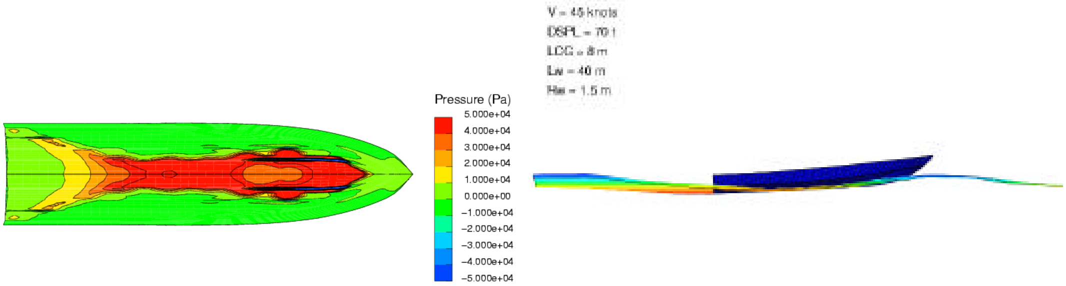
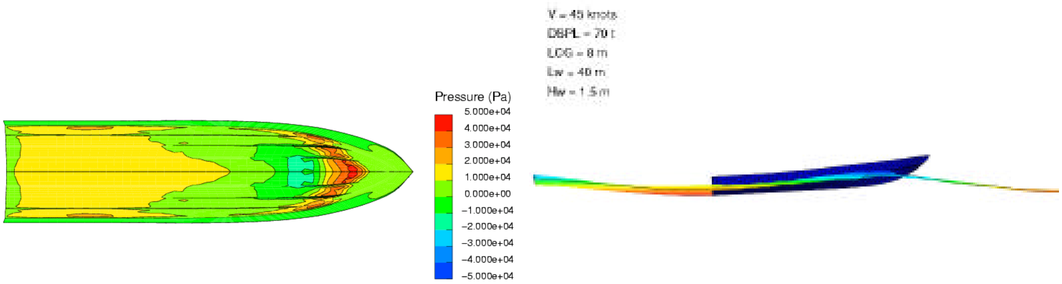


Figure 10

The next figures show the calculated pressure distribution, and the corresponding trim of the hull, at three different instants of the simulation. In the first frame be boat is nearly out of the water and there is only a small vee-shaped high pressure region ahead of the transom. After that the hull, falling down, hits the water all along the keel, and the impact pressure is maximum. The water is partly deflected by the spray rails. Finally the bow enters the incoming crest. Of course these data can be very useful for structural calculations.







3. Conclusions

In the paper the need of a tool able to predict the sea keeping performances of hard chine vessels has been pointed out. The method investigated to face the problem if an unsteady CFD solver where the free surface is calculated using a VOF method. A code has been developed to allow the movement of the structured mesh with time. The boat is free to trim and sink, under the effect of regular bow waves generated at the inlet. The results obtained in the first tested case are still to be validated, but the code has been proven to be robust and supply qualitatively satisfactory results.

4. References

1. CAPONNETTO, M., "*Numerical Simulation of Planing Hulls*". Proceeding of the 3rd Numerical Towing Tank Symposium, 9-13 September, Tjärnö, Sweden.

2. CAPONNETTO, M. "*Practical CFD Simulations for Planing Hulls*" . HIPER '01, 2-5 May, Hamburg, Germany.

3. FERZIGER, J, H, PERIC, M, "*Computational Methods for Fluid Dynamics*". Springer editions.

4. COMET version 2.00, User Manual.

5. BERTRAM, V. "*Practical Ship Hydro Dynamics*". Butterworth-Hainemann editions.