Numerical Simulation of Planing Hulls

Mario Caponnetto, Rolla SP Propellers SA, Switzerland

The present paper deals with the numerical simulation of the flow on hard chine planing vessels. In the past it has been difficult to study this kind of vessel by means of the available methods, such as panel methods, usually applied to conventional ships. The reason is due to the particularly complicated flow field generated by planing hulls.

A thin layer of water (spray) is generated at the stagnation line, and in this zone the pressure gradient is very high. Depending on the hull shape and speed the flow may separate sharply at the chine, and eventually reattach along the hull sides, or generate vortices. The transom can be dry at high speed, while partly wetted, with recirculating water, at lower speed. The generated bow and stern waves are normally steep and may break, dissipating their content of energy.

Due to these particular features, potential flow methods are not suitable for planing hull computation, nor are any methods that are unable to compute highly geometrically nonlinear free surfaces.

The method presented in this paper approaches the problem using a RANS code based on Finite Volumes, where the free surface is computed using a Front-capturing Method (Volume of Fluids).

The results obtained using this method for real hull shapes and monoedric hulls are very satisfactory; some examples are presented in the paper.

Introduction

Hard chine planing hulls represent nearly the totality of the vessels employed in the nautical field for pleasure crafts, and are also largely used for small and medium size civil and military applications.

Despite this fact there is a lack of tools at disposal of the naval architect to design, analise and optimize this kind of vessel. The analytical method normally used to predict the resistance and the trim of the boat is the well known Savitsky method, with its variations. The main drawback is that the Savitsky method is strictly applicable only for monoedric hull shapes, namely any variation of the beam and deadrise angle cannot be properly taken into account. The formula are a mixture of experimental and theoretical data and some of the approximations adopted may become important. For example the effect of the hydrostatic pressure and the position of the center of pressure are a too simple approximation of the real phenomenon. In general large errors may occur for warped hulls and when computing performances at low speed.

There are also a number of systematic series, but again the shape of the real planing hulls used nowadays normally differs from that of the series.

Moreover it is not a common practice of the majority of the shipyards to perform model testing in the towing tank, and this is mainly due to the obvious time and cost constraints typical of small boats.

All these factors make the prediction of planing boat performance more an art then a science, and the result is a large margin of uncertainty.

The need for a reliable direct method able to compute the flow field around planing hulls is then highly requested by designers, but the mathematical complications due to the complexity of the flow are obvious. This is due to the presence of the spray, detachment of the flow at the chine and reattachment along the side, complicated flow at the transom and presence of breaking waves.

A number of recently developed CFD codes are now able to compute flows with large deformations of the free surface. Particularly successful are those adopting the Volume of Fluid technique. In this case both air and water are computed simultaneously; the two fluids are not considered separately, but as a continuum fluid with different physical properties. The physical property of the actual fluid depends on the properties of the two constituent fluids and a scalar quantity (C) that represents the volume fraction of the two phases (i.e. C=1 for water and C=0 for air). A value of C between 0 and 1 indicates the presence of the interface, that is not discontinuous but mainly depending on the mesh size.

A similar approach is used in the commercial code Comet, developed at the ICCM of Hamburg. The code is a Finite Volume method able to solve RANS equations with the additional above mentioned capability to include free surface calculation.

The code has been used to compute a large number of planing hulls and the comparison with the available experimental data has been very satisfactory. Accurate resistance curves for different displacements and center of gravity positions can now be computed in a couple of days, with obvious advantages for the boat (and propeller) designers.

Mesh and test procedure

An accurate and efficient mesh is needed to compute the flow field with the requested precision and within practical time constraints. While Comet allows both structured and unstructured mesh, for our problem we use multi-block mesh with structured hexahedral cells. Nine blocks are normally used to define the computational domain, distributing the cells between the blocks in order to refine the mesh where the gradients are expected to be higher. Cells are refined along the hull surface to capture the spray and the boundary layer, as well as above and below the undisturbed water surface to compute waves. Cells do not have to match at block interfaces. A typical mesh is represented in figure 1.

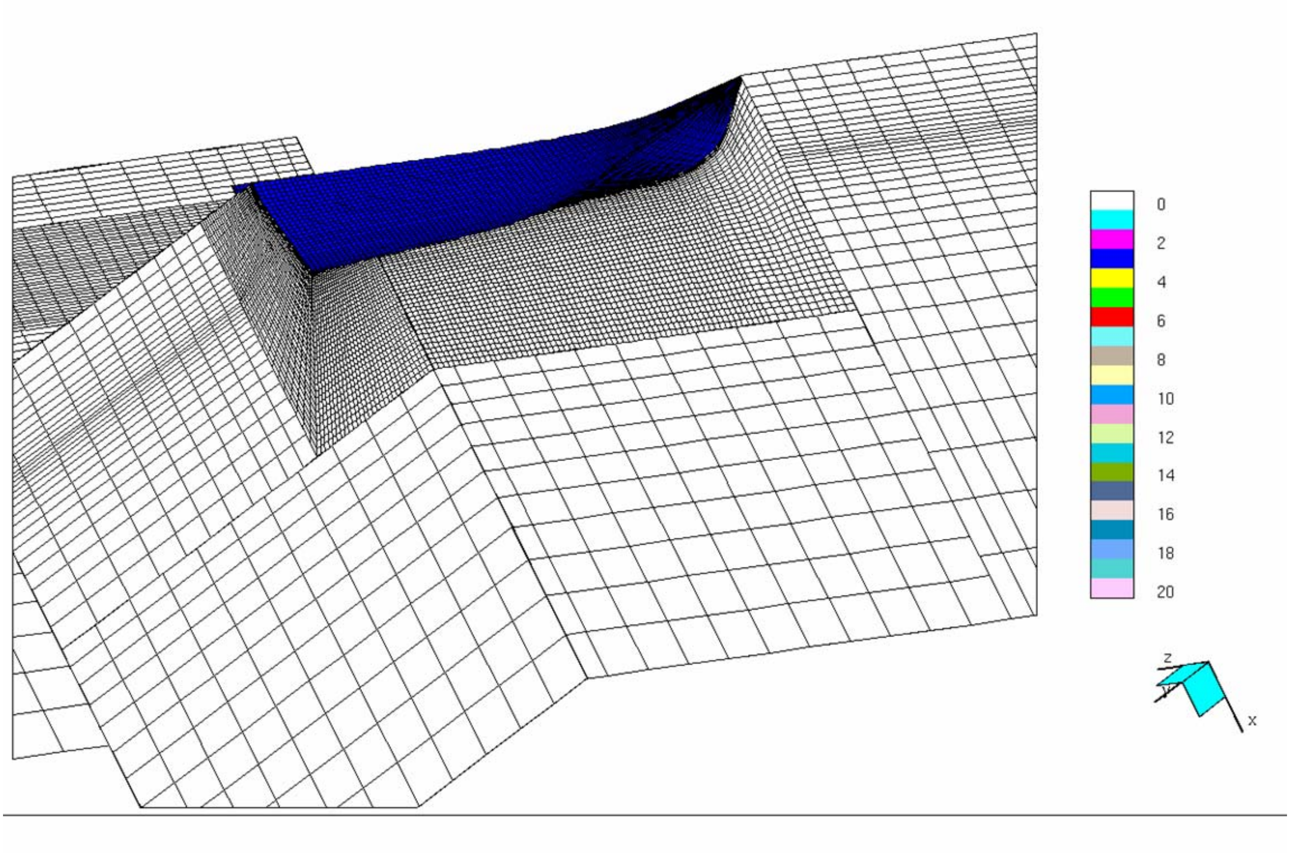

Figure 1

After a number of convergence tests we have noticed that, while the flow detail increases with the cell number, a mesh having about 400.000 cells (for one half of the boat) is sufficient to have converged values of the forces acting on the hull (lift and drag) and of the center of pressure.

Given the hull shape, displacement, center of gravity and speed, the trim of the boat remains unknown. In theory we should test different trim (and submergence) until we obtain, with a trial and error procedure, the trim that satisfies the equilibrium of the forces and moments. Since the displacement of the boat may vary and moreover we could be interested to know how the resistance changes moving the center of gravity, we prefer to test the boat at a number of predefined trim and submergence, to compute the lift, resistance and the center of pressure, and then to interpolate these values at the desired displacement (lift) and center of gravity (center of pressure).

In general for each speed the boat is tested at a combination of 3 trim angles and 3 immersions of the transom. The results of the 9 tests allow later a bi-quadratic interpolation of the values. While Comet can perform parallel computation, in our case it is has been found convenient to run the cases serially, each one on a different processor that we have at disposal (2 processors of SGI Octane, 4 processors of SGI Origin 2000).

Figures 2 and 3 show some typical result; in this case the data are for a 80 foot boat at 34 knots. Figure 2 shows how the position of the center of pressure (from transom) and the lift change with trim and submergence. Each curve is for a different trim angle (2, 3 and 4 degrees), while each point over the curve is a different submergence at the transom. Figure 3 shows similar results for the total resistance.

In many cases not only the value of the forces are of interest, but also the pressure distribution and the wave pattern visualization can be useful to understand the behavior of the boat. Figure 4 shows the free surface elevation calculated for a boat with unusual hull shape running at 40 knots. It can be observed the rise of the spray at the stagnation line; the spray is then deflected at the chine and moves sideway generating a thin breaking wave. The level of the free surface is taken conventionally where the volume fraction is C=0.5.

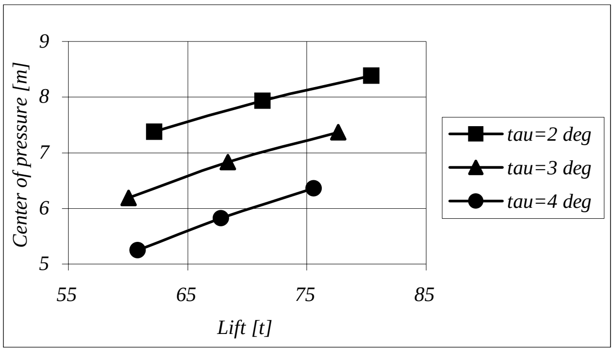

Figure 2

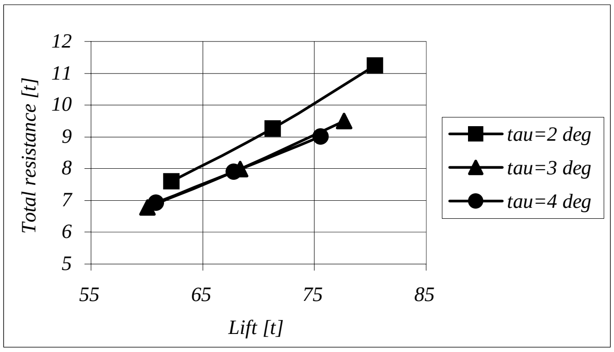

Figure 3

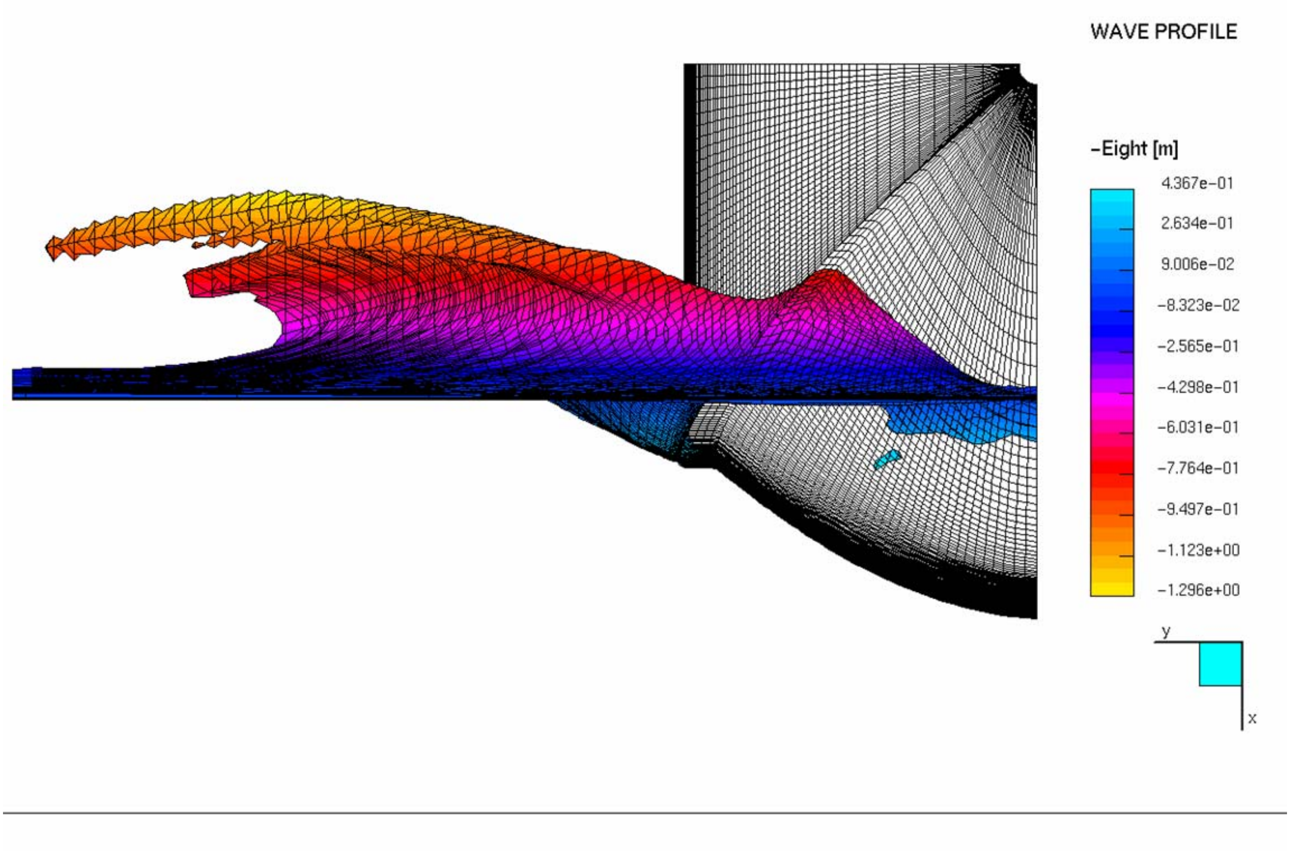

Figure 4

Validation of the method

A number of computations have been performed for comparison with the available experimental data. In particular the code has been extensively compared with the Savitsky method in a wide range of speeds and for different geometries. Figure 5 and 6 show one of these comparisons for a monoedric hull having 12 degrees of deadrise angle. The hull is fixed at a given trim and submergence and the speed is increased from 25 to 80 knots. As expected the lift increases with speed more or less quadratically, while the center of pressure moves forward reaching asymptotically the limiting value of 0.75Lm predicted by Savitsky method. The lift is overpredicted, but the difference between the two methods decreases with increasing speed.

As already mentioned 400.000 cells are normally used for hull computation; this figure has been chosen as a compromise between precision and computational time constraints. In order to see the effect of the mesh size, figure 7 shows the longitudinal lift distribution calculated for the same (warped) hull but using different meshes (200.000 and 700.000 cells). The two results are very close, but the finer mesh supplies a higher lift (+4% in this case). In general this difference goes to about 1% when using the mesh with 400.000 cells.

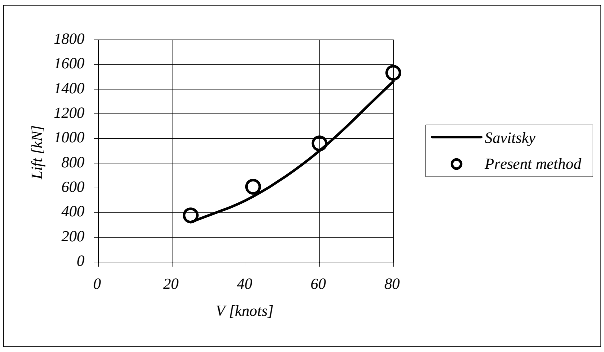

Figure 5

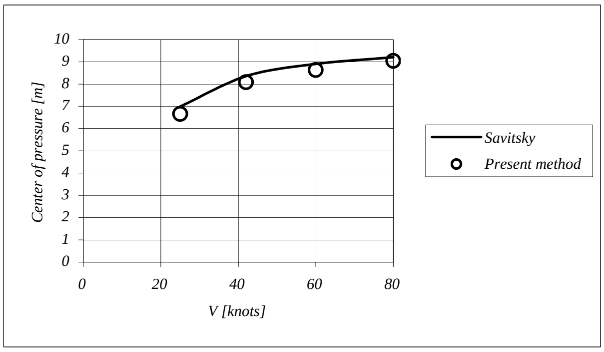

Figure 6

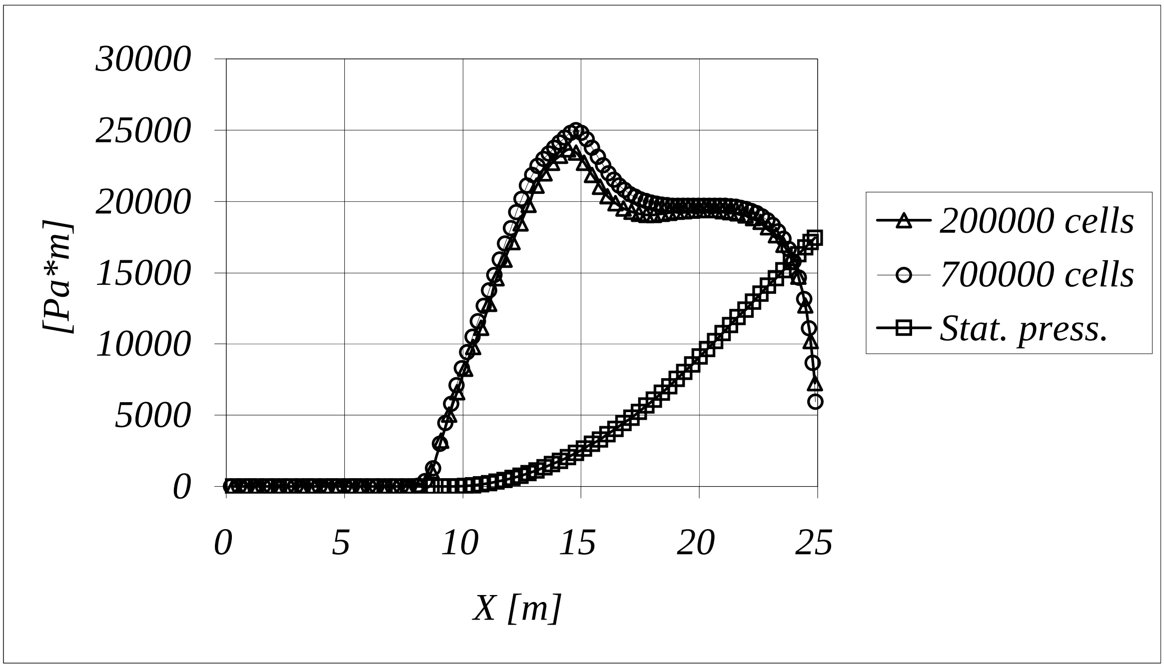

Figure 7

Figure 7 shows that the lift increases steeply from the point of attachment of flow at the keel (X=8 m) to the point of detachment at the chine (X=15 m). Then the lift decreases, going rapidly to zero close to the stern, since in this case the transom was completely dry.

Conclusion

In this paper has shown the possibility to compute planing hull flow using a numerical method. The method used is based on the Finite Volume formulation and the free surface deformation is computed using a Volume of Fluid approach. With this technique it is possible to predict the complex behavior of the flow typical of planing hulls (spray generation, detachment and reattachment of the water at the chine, breaking waves). The computed forces are in general in good agreement with the experimental data available, and some comparison is presented in the paper. Any realistic hull shape can be computed, and this is a great improvement for planing hull designers, compared with the methods commonly used in the past.