Practical CFD simulations for planing hulls

Mario Caponnetto
Rolla Research , Via Silva 5, Balerna (Switzerland), info@rolla-propellers.ch

Abstract

The flow of planing hulls is characterized by a large spray, a dry transom at design speed, sharp separation of the flow at the chine with eventual reattachment along the sidewalls. Large pressure gradients are experienced at the stagnation line. This behavior prevents the use of typical "panel methods", normally used to compute conventional ships at low speed. In this paper the application of a CFD code for the analysis and the design of planing crafts will be presented.

1. Introduction

Planing crafts represent nearly the totality of the small and medium size pleasure boats and a big percentage of military patrol vessels. Despite this fact and the obvious enormous market involved, there is a lack of methods able to help the naval architects in the design of planing boats. Normally they use simple theories, such as the Savitky method, or systematical series that are in many practical cases obsolete. Moreover the expenses of towing tank analysis can be seldom justified for a small boat.

The major drawback of the Savitky method is that it is strictly valid only for monoedric hulls; any variation of the deadrise angle and beam cannot be taken into account, while real planing hulls are normally warped. The simple formula used to compute lift and position of the center of pressure are a mixture of empirical and theoretical data, that in some cases are a too crude approximation of the real phenomenon. Results become more and more questionable as the speed of the boat decreases, since the effect of the hydrostatic pressure is considered in a very approximated way.

The systematical series available in the literature are in some extent obsolete, since they represent the typical hull shape of planing boats used forty years ago. They can be useful to analyze the main parameters of the hull, such as the length/beam ratio, but may other important form parameters cannot be analyzed.

In the last few years a number of codes based on the potential flow theory have been developed for the computation of planing hulls. Even if any kind of hull shape can be computed with these methods, the drawback is that they are only valid in the very high speed range, and are not suitable to compute performances at the hump and intermediate speed.

From what stated above it is clear that the design of planing hulls is more an art then a science. Shipyards design boats basing on their feeling and experience, with small changes from a boat and the next one. In many cases this pragmatic procedure works well, but sometimes the results are catastrophic. It is not unusual the case of boats designed for high speed that are not able to overcome the hump speed.

On the contrary of displacing ships, where the resistance curves are very steep, planing hulls, above hump speed, have a flatter resistance curve; this means that larger differences of speed can be gained or lost with minor changes of the hull or propeller efficiency.

Nowadays propeller designers have reliable tools able to optimize the propeller once the design conditions are known, but this is a little improvement if no sure resistance or wake data are available.

For these reasons we have considered the possibility to use the state of the art CFD codes to compute performances of planing boats. The first results obtained have been satisfactory, considering the peculiarity of the flow and the velocities that we are facing.

Now, at Rolla SP Propellers, it is a common practice to ask to the shipyard the geometry of the hull for whose we have to design the propeller. This allows to perform in few days a “numerical towing tank analysis”, obtaining the trim, the resistance curves and the wake field needed for the proper design of the propellers. Moreover it is possible to give to the designer useful suggestions for the improvement of its hull.

2. Overview of the numerical method

The capabilities of the commercial code Comet, developed at the ICCM, have been considered suitable to compute flows with large deformations of the free surface, typical of planing boats.

The code uses a classical Finite Volume Method (FVM) to solve the Navier-Stokes equations, and a K-epsilon model is adopted for turbulence modeling. The special feature of the code is the Front-Capturing Method, used to compute the free surface. This requires a sort of two-phases method; in our case air and water are considered as a single fluid in the whole domain, whose physical properties depend on the concentration of the two constituent fluids (fluid 0 air and fluid 1 water).

Calling volume fraction C the concentration of the fluid 1 in the fluid 0, we have for the density and the viscosity:

$$\rho = C\rho_1 + (1-C)\rho_0$$
$$\eta = C\eta_1 + (1-C)\eta_0$$

Moreover the mass fraction c of fluid 1 is given by:

$$c = C\frac{\rho_1}{\rho}$$

In regions of the domain where the value of C is between 0 and 1, fluid 0 and 1 share the same pressure and velocity.

The numerical implementation of this method (the High Resolution Interface Capturing scheme HRIC is used to compute the convective transport of the scalar quantity) introduces some amount of numerical diffusion; moreover the interface between the two fluids has always a finite transitional area that is dependents on the accuracy of the mesh used. For convention we consider the free surface at the point where C=0.5.

The method allows computation of very complex free surfaces and in particular is able to represent thin sprays, breaking and overturning waves, inclusion of air bubbles in the water (ventilation).

The domain is initialized filling of water all the cells below the undisturbed free surface and setting the initial velocity equal to the boat speed. The solution is solved marching in time, using a first order implicit Euler discretization, and the evolution of the flow is followed until a steady solution is found.

In general a time step corresponding to a Courant number equal to 0.25-0.5 is used.

Since we are looking for the stationary solution, only one iteration is accomplished for each time step.

3. Mesh

One practical advantage of this code is a certain freedom in the generation of the mesh near the free surface, at condition that, in order to find a sharper interface, enough small cells are located in the transition region between the two fluids. The computational domain is devised in blocks; in general, at the interface between the blocks, vertices are not matching. Clearly cell density is set higher in the region surrounding the hull (particularly in the boundary layer and spray region) and close to the free surface. Moreover cells should be smaller approaching the transom, to capture the rise of velocity consequent to the establishment of the Kutta condition. A typical mesh is shown in figures 1 and 2.

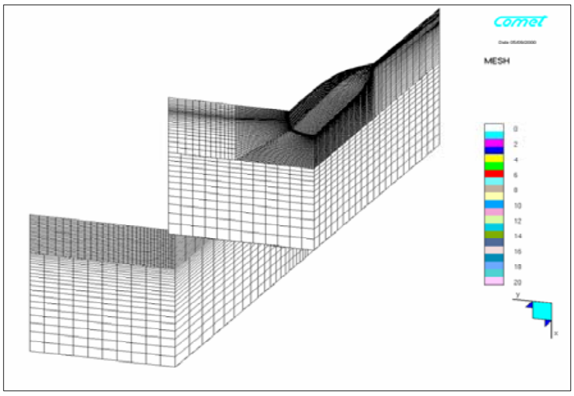

Figure 1

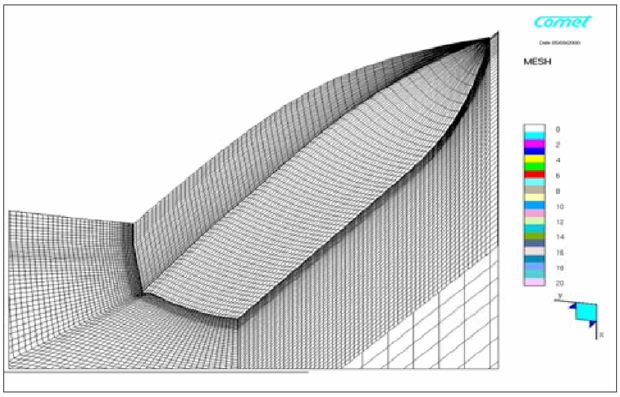

Figure 2

Normally the total number of cells used range between 300.000 and 800.000. For practical computations, we have seen that 500.000 cells are a good compromise between accuracy and computational time. More cells should be used when the hulls have one or more spray rails, since pressure and velocity gradients are locally very large. It is the case of the boat in figure 3, that shows the mesh arrangement of a large planing hull having four spray rails and a large tunnel to fit a large diameter propeller.

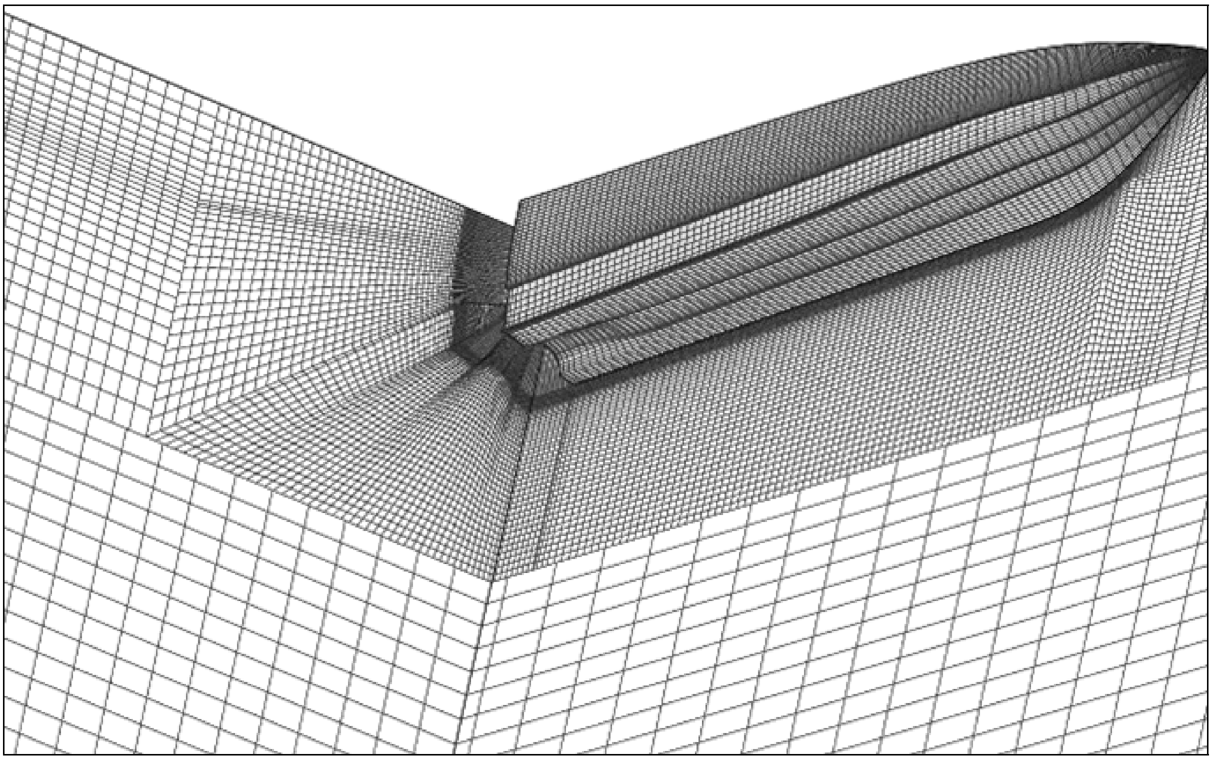

Figure 3

4. Present method versus Savitsky

A number of computations have been performed to check the accuracy of the present method versus the empirical data available from Savitsky method. It must be pointed out that Savitsky is only a "statistical" method valid for prismatic hulls; it is quite accurate for high speed, but its accuracy decreases with decreasing speed.

Two prismatic hulls, having respectively 0 and 15 degrees of deadrise angle and 0.3 m of beam at the chine, have been tested at fixed trim (4 degrees) and transom draft (0.045 m), varying the speed.

Four speeds have been tested, corresponding to speed coefficients Cv equal to 0.6, 1.5, 3.0 and 6.0 (Cv=V/(gB)^.5).

For each case two different meshes have been checked, one very coarse with about 85.000 cells and another one finer with 230.000 cells.

4.1 Lift and center of pressure

The results are summarized in tables 1 and 2. LCP is the distance of the center of pressure from the transom. Pressure drag is not presented since for a prismatic hull it is simply the lift for the tangent of the trim angle.

Lift and position of the center of pressure are calculated following Savitsky with the formula:

$$L = \tfrac{1}{2}\rho C_{L\beta} b^2 V^2; \qquad LCP = \lambda b \left[0.75 - \frac{1}{5.21\dfrac{C_V^2}{\lambda^2} + 2.39} \right]$$

$$C_{L\beta} = C_{L0} - 0.0065\beta C_{L0}^{0.60}; \qquad C_{L0} = \tau^{1.1}\left[0.012\lambda^{1/2} + \frac{0.0055\lambda^{5/2}}{C_V^2} \right]; \quad \lambda = \frac{\left[\dfrac{d}{\sin\tau} - \dfrac{b}{2\pi}\dfrac{\tan\beta}{\tan\tau} \right]}{b}$$

$b = beam\ at\ chine$

$d = transom\ draft$

$\beta = angle\ of\ deadrise$

$\tau = trim\ angle$

The results are also plotted in figures 4 and 5. It can be seen in figure 4 that the agreement on lift between Savitsky and the present method becomes better increasing the speed.

At low speed the lift predicted by Savitky seems to be underestimated. For the 0 deadrise hull the lift should go to the hydrostatic value of 43.7 N, while Savitsky predict only 32.2 N. While the formula are valid for Cv greater then 0.6, we still image that lift is underestimated on a wider range of speed.

The agreement between the two methods is better for the 0 deadrise then for the 15 deadrise hull.

Figure 5 shows the longitudinal center of pressure. The two methods give similar results; the LCP should move asymptotically from the hydrostatic value to 75% of the "mean" wetted length. This trend is clear, but our computation overpredicts LCP for the 0 deadrise hull, while underpredicts LCP for the 15 deadrise hull.

It is noticeable to see that the very coarse mesh gives results very close to those obtained with the finer mesh.

Deadrise angle = 0 degrees

	85.000 cells		230.000 cells		Savitsky	
Cv	Lift [N]	LCP [m]	Lift [N]	LCP [m]	Lift [N]	LCP [m]
0.6	45.26	0.2667	…..	…..	36.45	0.2786
1.5	73.65	0.4041	72.20	0.4036	58.58	0.3810
3.0	148.54	0.5083	146.16	0.4920	137.60	0.4780
6.0	468.86	0.5570	457.83	0.5470	453.68	0.5282

Deadrise angle = 15 degrees

	85.000 cells		230.000 cells		Savitsky	
Cv	Lift [N]	LCP [m]	Lift [N]	LCP [m]	Lift [N]	LCP [m]
0.6	....	....	....	....	11.25	0.2008
1.5	31.42	0.2684	31.37	0.2691	23.45	0.2828
3.0	82.33	0.3140	82.82	0.3145	68.14	0.3249
6.0	276.07	0.3320	260.15	0.3170	247.29	0.3401

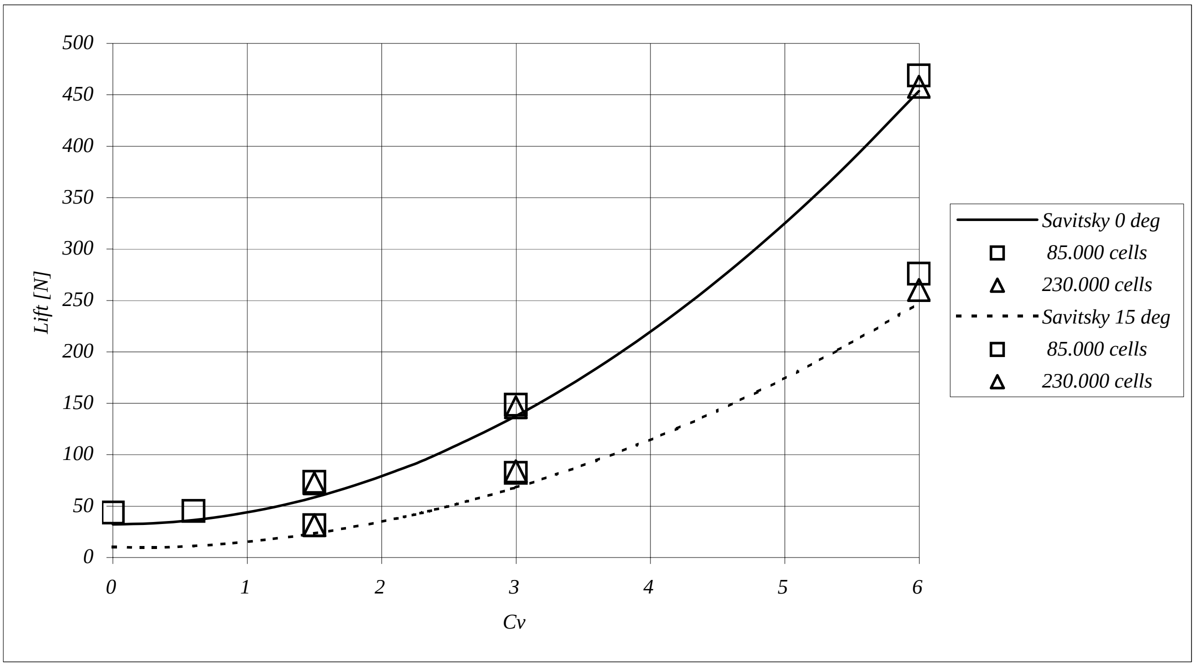

Figure 4

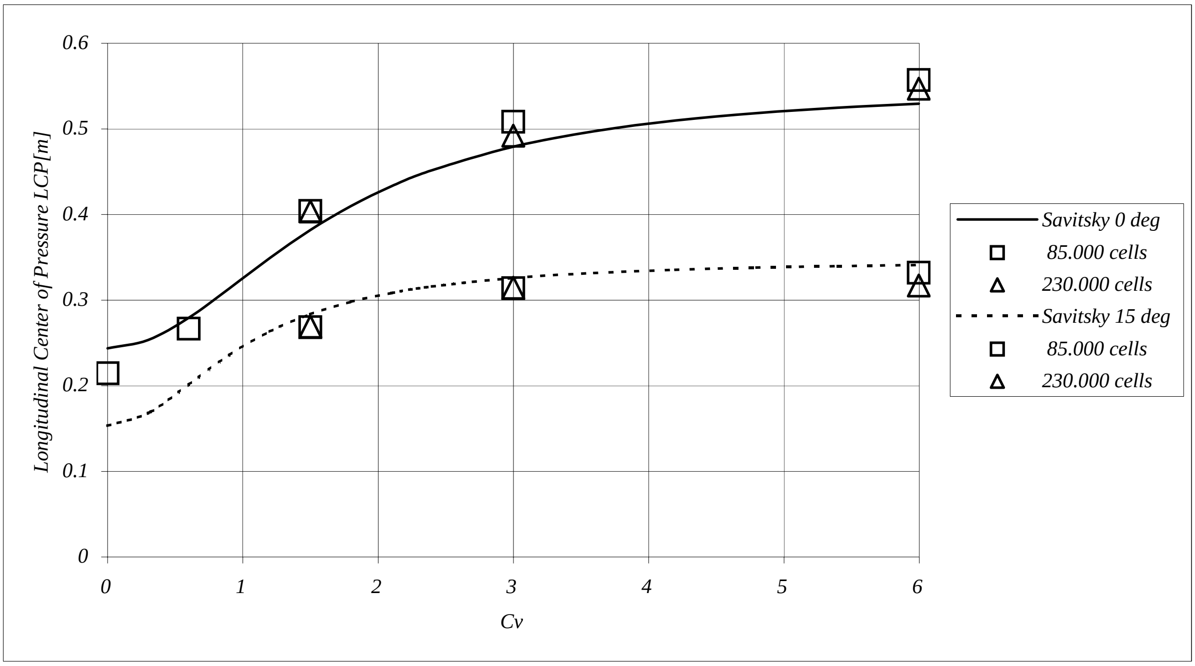

Figure 5

4.2 Free surface profiles

The calculated wave profiles for Cv=1.5, 3 and 6 and deadrise angles equal to 0 and 15 degrees are shown in figures from 6 to 11.

At Cv=1.5 the volumetric Froude number is about 2. Both hulls (figures 6 and 9) are already in the planing region but the free surface profile still resembles that of a displacing hull at high speed.

There is a divergent wave (partially breaking) at the bow; the flow sharply detaches at the transom, rising rapidly downstream.

Similarly the water flowing from the sidewall tends to move toward the centerline, moving behind the transom. The two flows coming from the bottom and from the sidewall reattach forming a steep crest (rooster tail) that moving further downstream spreads laterally forming a divergent wave.

Going to Cv=3.0 (figures 7 and 10) the volumetric Froude number is about 3.5. The free surface profile is qualitatively similar to Cv=1.5 but the waves are stretched longitudinally.

In particular the flow coming from the bottom of the transom and from the sidewall need more distance to reattach and to form the rooster tail.

At Cv=6.0 (figure 8 and 11) we are in the very high-speed region (volumetric Froude number around 6). There is no presence of the “classical” waves.

Only the steep crest formed at the stagnation line and a long and deep hollow behind the transom are visible. For the hull with 0 deg deadrise the pile-up of the crest is nearly vertical, while for the 15 deg deadrise case the crest is overturning. This phenomenon is visible in figures 12 and 13, were different transversal cut are presented.

Figure 14 shows the same case (15 deg deadrise and Cv=6) computed with a finer mesh having 580.000 cells. The behavior is qualitatively similar, but the finer mesh is able to capture the steepness of the overturning wave. The increased accuracy of the mesh does not affect significantly the values of the computed lift and center of pressure.

In figures 13 and 14 it clearly shown the formation of the spray.

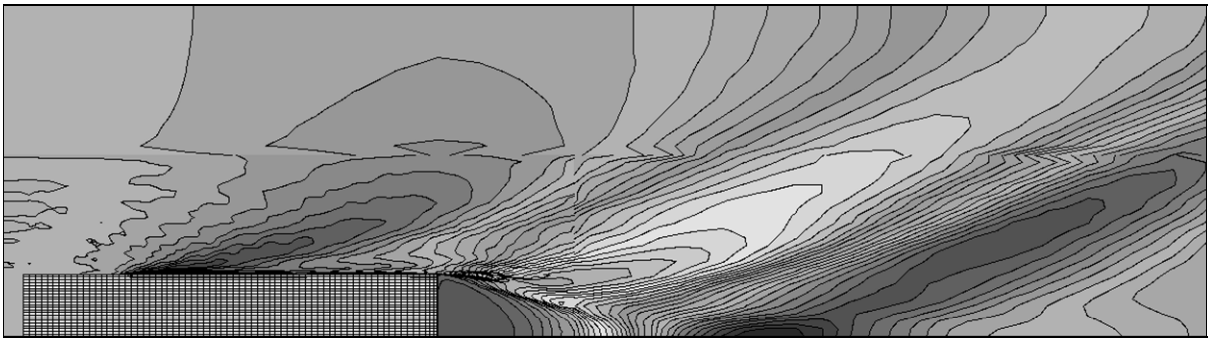

Figure 6
Deadrise angle =0 deg; Cv = 1.5

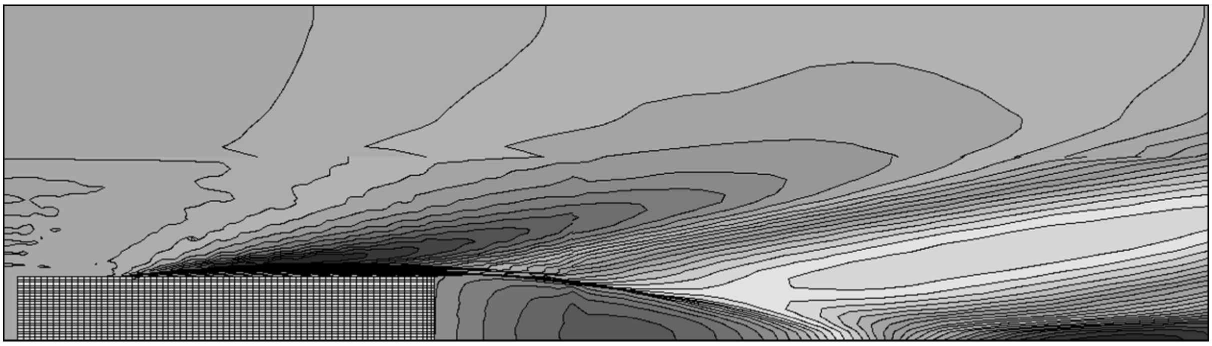

Figure 7
Deadrise angle =0 deg; Cv = 3.0

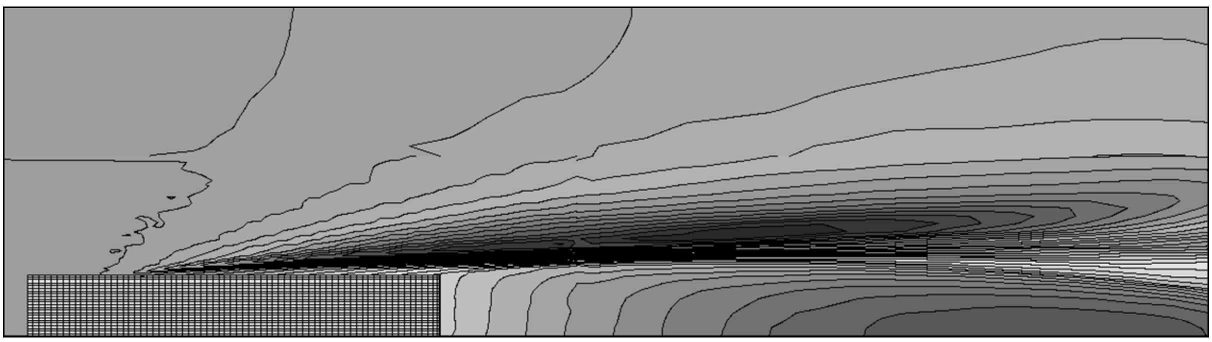

Figure 8
Deadrise angle =0 deg; Cv = 6.0

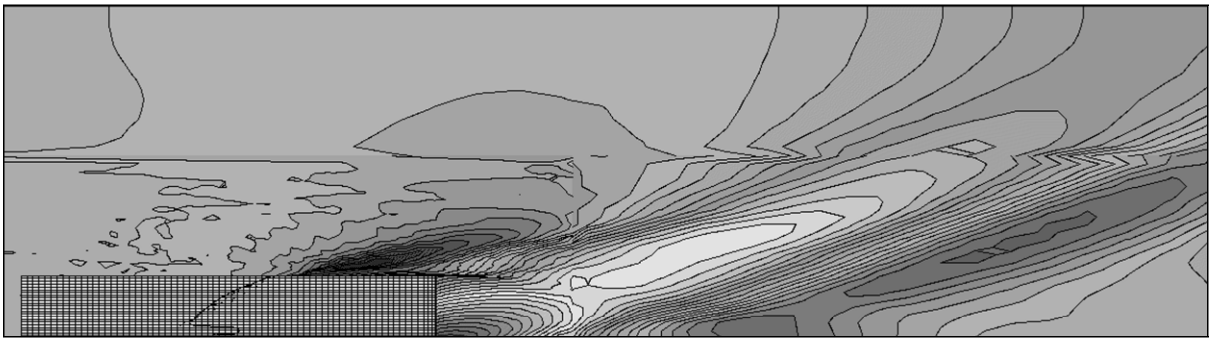

Figure 9
Deadrise angle =15 deg; Cv = 1.5

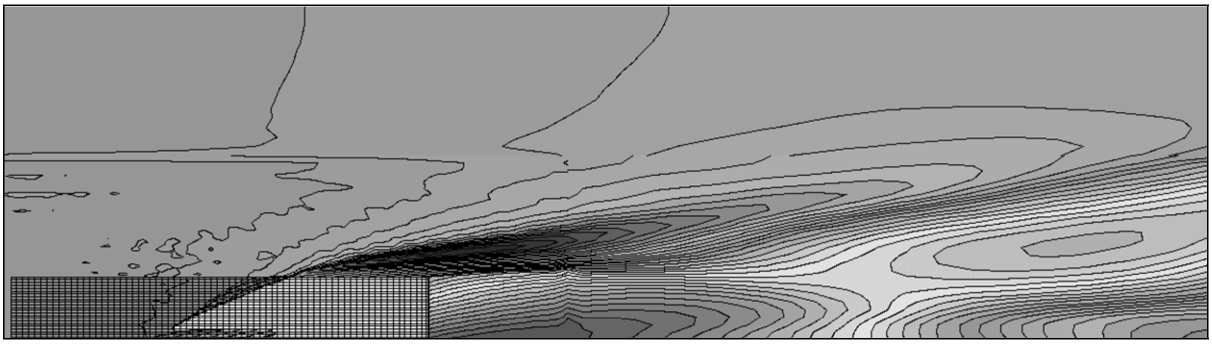

Figure 10
Deadrise angle =15 deg; Cv = 3.0

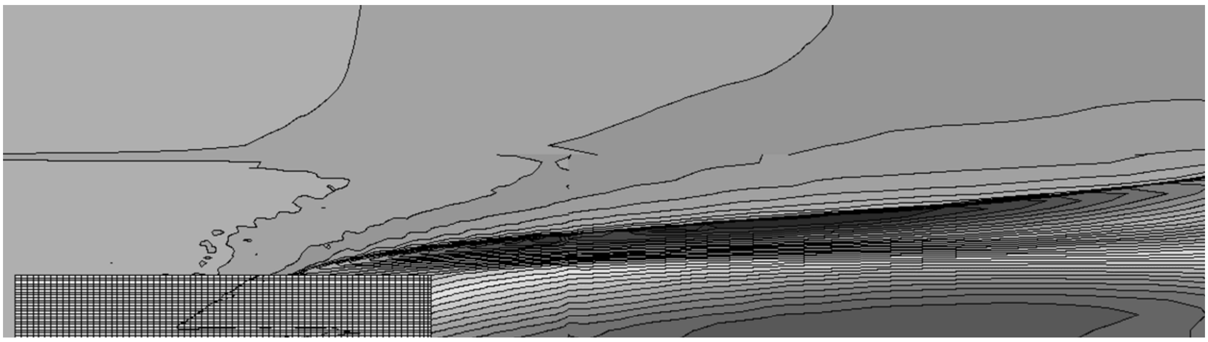

Figure 11
Deadrise angle =15 deg; Cv = 6.0

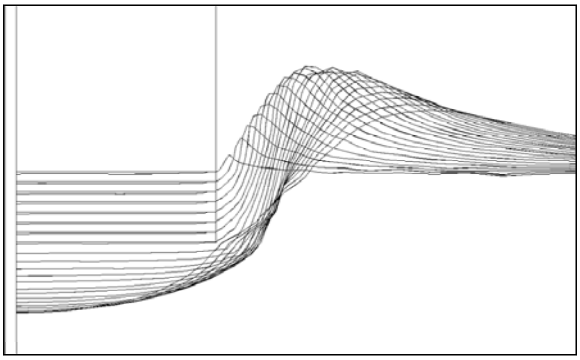

Figure 12
Deadrise 0 deg; Cv=6; 230.000 cells

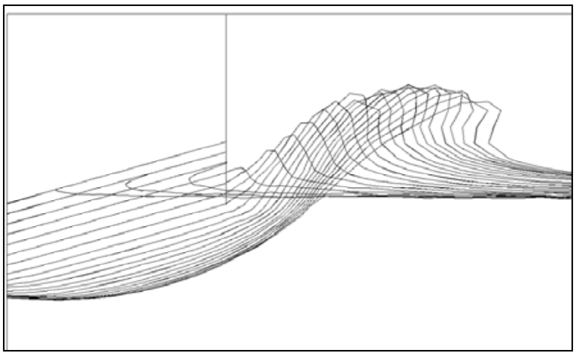

Figure 13
Deadrise 15 deg; Cv=6; 230.000 cells

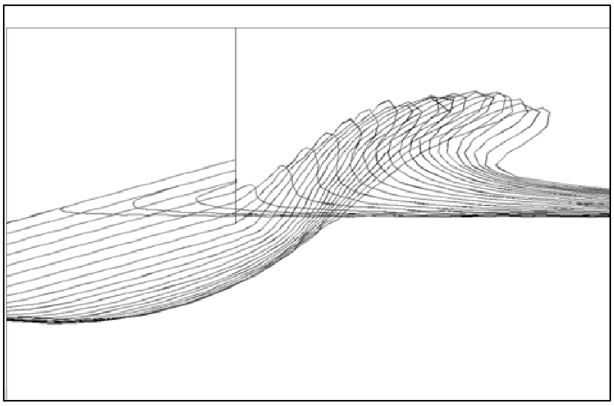

Figure 14
Deadrise 15 deg; Cv=6; 580.000 cells

5. Practical application of the method

The present method is daily used at Rolla SP Propeller for the computation of trim and resistance of planing boats. The reason, already explained in the introduction, is the typical lack of towing tank analysis and the fact that the design of propellers for this kind of boats requires not only the knowledge of the effective power of the boat, but also the effect that secondary forces developed by the propellers have on the attitude and the performances of the boat.

Moreover the use of the CFD can be greatly useful to understand the characteristic of the flow and to suggest to the shipyard improvements and corrections in an early design stage.

For a given boat and speed, with the hull set at a particular sinkage and trim angle, the lift and the position of the center of pressure are an output of the computation. In general, we should find the correct trim of the boat with a trial and error procedure. It is also true that the displacement and the position of the center of gravity of the boat are a question mark until the sea trials, and anyway they change during the life of the boat.

So we don't try to find results only for a particular condition of displacement and center of gravity, but for a reasonable "range" of these two values. In practice for each speed 9 cases are computed for a combination of 3 trim angles and 3 sinkages.

Once the lift, drag and center of pressure are computed for each case, it is easy to interpolate quadratically to the desired value of the displacement and center of gravity.

An example is shown here. The boat of the example was designed for a maximum speed of 50 knots and the estimated displacement and position of the center of gravity from the transom were respectively 80 tons and 9 m.

After a preliminary test with a very coarse mesh it was decided to test the boat at 3.0, 3.3 and 3.6 degrees of trim, and for each trim 3 different immersions of the hull. Figure 15 shows the computed lift and LCP for the 9 cases, while the corresponding total drag is shown in figure 16.

It is clearly shown in figure 15 that the estimated design condition falls inside the tested cases and moreover it is reasonable to interpolate values in a range of displacement from 75 to 90 tons and for LCG between 8 and 10 m.

Figure 16 shows that the total resistance increases decreasing the trim angle, namely moving the center of gravity forward. This is typical for this range of speed (volumetric Froude number around 4) since with the reduction of the of trim the increase of wetted surface and friction drag overcomes the reduction of pressure drag.

It is difficult, for practical reasons, to chose the desired position of the center of gravity and moreover, in this case, a movement of the LCG further aft could decrease the performances at lower speed or even to prevent the overcome of the “hump speed”.

On these basis it was suggested to test a modified hull obtained adding a small rocker, namely a convexity in the aft part of the hull. The low pressure generated by the rocker, while giving negligible effects at low speed, increased the trim angle of about 0.2 degrees, with a consequent reduction of drag of about 3%.

In the simplest way the hypothesis Lift=Displacement and LCP=LCG can be imposed, but especially when dealing with boats having surface piercing propellers this hypothesis can be too restrictive.

Once the percentage of lift developed by the propellers is know, it is very easy to compute the correct trim (and resistance) solving the right equilibrium of forces and moments acting on the boat. On the other side it is possible to “tune” the propeller to obtain the optimum trim.

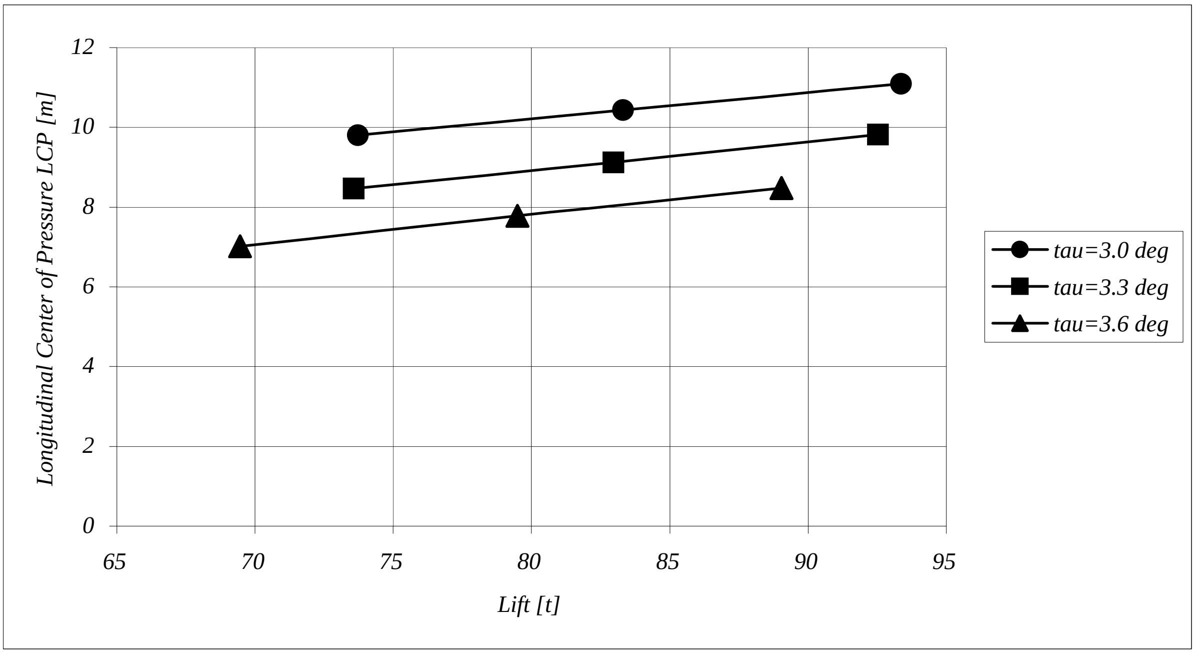

Figure 15

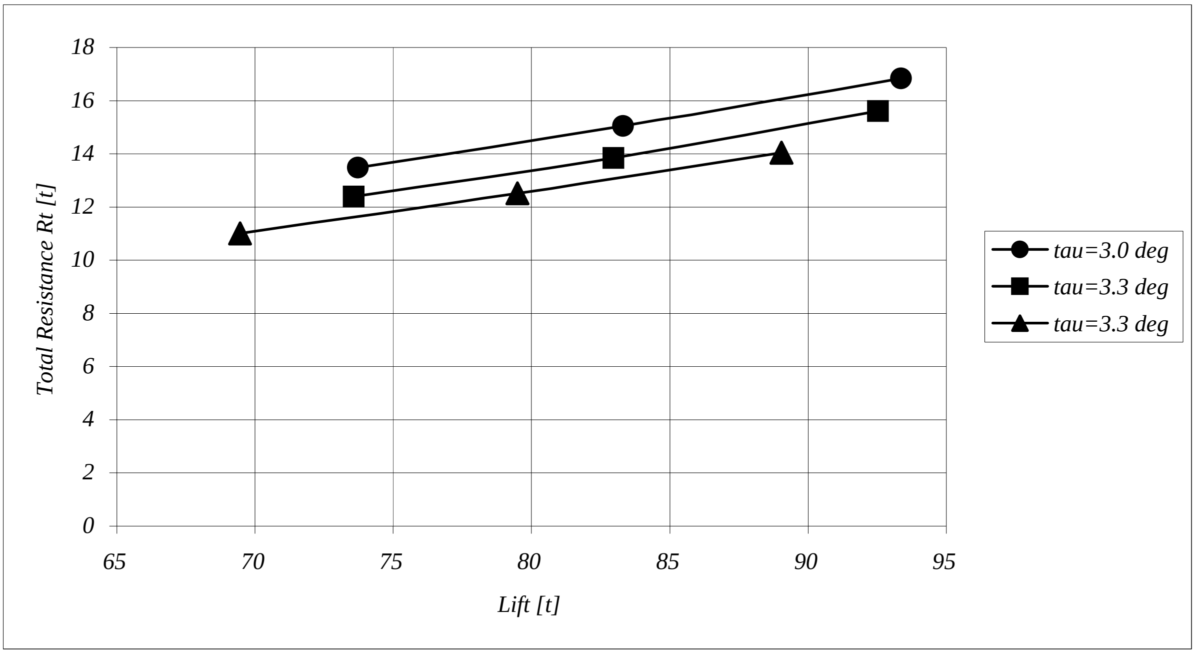

Figure 16

6. Conclusion

In the paper the capabilities of the proposed CFD code to deal with the computation of planing hulls has been shown. The methods is able to compute the large deformations of the free surface typical of fast boats. The comparison with the Savitsky method is in general acceptable. The advantage of direct computations lies in the possibility to analyze and compare “real” hull shapes, that are in general non monoedric.