# A NEW PROPELLER DESIGN METHOD FOR FAST PLANING BOAT APPLICATIONS

**Mario Caponnetto** *University of Genoa, Italy*
**Philip Rolla** *Rolla SP Propellers, Switzerland*
**Marzio Porro** *Rolla SP Propellers, Switzerland*

**ABSTRACT**
*A new method for propeller design is presented. It is based on the iterative use of a Vortex Lattice Method to obtain the optimum circulation, chord and thickness distribution, and a Panel Method to calculate the pitch and camber along the blade span. The optimisation is performed taking simultaneously into account viscous effects, cavitation constraints, structural aspects and the complete three dimensional geometry of the blade.*

*An application to the design of fast yacht propellers is presented; for this application a comparison is done between conventional NACA profiles and a new type of profile designed to reduce cavitation on propellers with high P/D ratios and inclined shaft.*

*Finally, the application of an unsteady Panel Method able to calculate the interaction between a couple of propellers and the overhanging bottom of a planing hull is presented.*

## 1 INTRODUCTION

The first step of any propeller design, for a given thrust (or engine power), RPM and advance velocity, is usually the choice of the main parameters such as the diameter of the screw, the chord, thickness and the optimum circulation distribution along the blade span.

Traditionally the problem is approached using at first the lifting line theory to obtain the optimum circulation distribution in inviscid conditions. The efficiency of the propeller is analytically written as a function of the unknown circulation distribution, and the maximum of this function is generally found using the Lagrange multipliers (Kerwin *et al.*, 1986). Then the chord length and thickness of each section is obtained from the requirement of cavitation avoidance and structural strength, using respectively two dimensional cavitation buckets (Brockett, 1966) and the cantilever beam theory.

In the second step of the design it is necessary to calculate the pitch and the camber of each section. The lifting line with lifting surface corrections (Morgan *et al.*, 1968), or the lifting surface code (Greeley and Kerwin, 1982) are usually utilised to solve this problem.

There are many drawbacks in this approach.

First, the blade geometry can be seldom adequately approximated with a "line", especially for fast propellers having large expanded area ratios. Namely, even considering the optimisation from the pure potential flow point of view, the optimum circulation calculated with the lifting line approximation may differ from that predicted once the skew, rake, pitch and actual chord length are included. This is due to the different interaction between free and bound vortices when placed in different spatial positions.

Second, there is a coupling between the circulation distribution and the corresponding chord and thickness required, and since chord length and thickness affect viscous drag, the optimisation of these three parameters must be done one at the time to find the "overall" optimum propeller in a viscous flow.

At last the calculation of the pitch and camber distribution with a lifting surface code is affected by some errors due to the approximation of the blade with its mean camber surface and the exclusion of the hub. This leads to a poor definition of the flow at the leading edge and at the inner propeller sections. Unfortunately these zones are responsible for most of the problems related to cavitation on fast propellers.

Due to the complex link between all these factors, and since other constraints can be part of the design, there is no way, from author's experience, to describe the efficiency of the propeller in a closed analytical form, nor to find the maximum of this function. A different way to approach the problem has been looked for.

The idea is based on a trial and error procedure; basically several different propellers, all satisfying the design requirements, are tested one at the time with an analysis method, calculating their efficiency. Each time a more efficient propeller is found, it is used as base for a new candidate optimum propeller, modified via a small random perturbation. While the idea could seem impracticable, there is a way to minimise the tentatives to perform and to converge, in a brief CPU time, to the overall optimum propeller.

The code developed to solve this problem is formed by two main routines, called the Optimisation Routine and the Design Routine; an external loop allows the repetition of the Optimisation and the Design Routines the number of times required to reach the convergence.

The Optimisation Routine is essentially a Vortex Lattice Method. The input are the design data and the best approximation of the blade and trailing wake available from the previous iteration; the corresponding grid of vortices is then generated. Several different circulation distributions are tested, distributing the proper vorticity on the grid and calculating the inviscid forces. For each circulation tested, the chord and thickness distribution that minimise cavitation and verify strength is chosen; then the efficiency, including the viscous forces, is calculated. After many tentatives the propeller with the maximum efficiency is temporarily selected.

The Design Routine is based on the Panel Method (Morino and Kuo, 1986, Hsin *et al.*, 1991 and Caponnetto *et al.*, 1994). The input is the best approximation of the blade (hub) and wake coming out from the Optimisation Routine, plus the corresponding best circulation distribution. Several pitch and camber distributions are tested until the prescribed circulation distribution is obtained. In addition a condition at the leading edge is imposed; in the paper we will consider the case of "shock free" propellers. Going out from the Design Routine, the trailing wake shape is updated with alignment to the calculated stream lines. The output geometry of the blade (and wake) is the input for the next run of the Optimisation Routine or, at convergence, the optimum propeller geometry.

A sample design case is presented in the paper. A fast planing hull application is proposed, where the propeller has a relatively large diameter and low RPM. Two different type of profiles are compared, namely the conventional NACA 66 and a new profile specially developed to reduce cavitation for this kind of application.

While the design procedure is based on a stationary calculation, averaging circumferentially the oncoming wake field, one has to verify the behaviour of the propeller in the real unsteady conditions. For the propellers we are talking about the main source of non stationarity is due to the inclination of the shaft. A non stationary Panel Code has been developed; it is able to analyse in the time domain the potential flow field of a couple of propellers having an inclined shaft and an overhanging hull.

## 2 THE OPTIMISATION ROUTINE

The kernel of the routine assumes a given blade outline. Following the traditional Vortex Lattice Method, the blade is represented as a grid devised in M span wise and N chord wise vortices, plus a trailing wake of given shape. Using the Biot-Savart law it is possible to calculate the velocity induced between each couple of vortices, assumed of unitary circulation. This arrangement of vortices, and the corresponding influence matrix of the induced velocities, will be kept *unchanged* all the time inside the routine.

We start assuming an arbitrary circulation distribution, namely a vector containing M values of $\Gamma_i$, one for each section. The circulation $\Gamma$ is distributed along each chord according to a desired law, for example the NACA a=0.8. Once each vortex has a defined circulation it is easy, utilising the influence matrix, to calculate the induced velocity on each vortex and then the inviscid thrust and torque.

While, as already mentioned, the geometry of the vortex lattice used to calculate the inviscid forces is kept frozen, one has to calculate, for the $\Gamma$ vector we are considering, what should be the best chord and thickness distribution that minimise cavitation and verify strength. This calculation is performed sequentially for all the M sections in the following way. The sectional bending moment is know from the integration of $\Gamma$ along the span. The centrifugal force at each section is known too since the calculation begins at the tip of the propeller, and hence the weight of the blade portion outer of the given section is known. Once a class of profiles is selected, for example the NACA 66, knowing the operating condition of the section during the blade revolution, due for example to the inclination of the shaft or due to a non uniform wake field, it is straightforward to calculate the chord and thickness that minimise the cavitation and satisfy the admissible tension.

It is now possible to calculate the viscous drag and to correct the thrust ($T$) and the torque ($Q$). In general for an arbitrarily assumed vector $\Gamma$ the design thrust ($T_{design}$) or torque ($Q_{design}$) won't be satisfied. Then the above calculation is repeated many times with new $\Gamma'$ distributions, obtained from the previous $\Gamma$ according to the simple relationship:

$$\Gamma_i' = k\Gamma_i \qquad j = 1,\dots,M$$

$$k = \frac{T_{design}}{T}, \text{ or } k = \frac{Q_{design}}{Q},$$

**Equation (1)**

hence preserving the adimensional circulation distribution shape, until convergence. The efficiency of the propeller $\eta$ can now be calculated.

The second step is the repetition of this calculation but starting from a new $\Gamma$ vector, obtained from the previous one via the addition of a "small random" $\Delta\Gamma$ vector. As above, the design thrust or torque respect is forced using Equation (1), and the efficiency of the new propeller is calculated. This is compared with the efficiency of the previous propeller; the propeller of the two with the lowest efficiency is discarded while the other one is used as base for a new random perturbation.

Going ahead in the iterations, any new variation will always start from the circulation distribution ($\Gamma_{opt}$) that showed the highest efficiency ($\eta_{opt}$) among all the previous iterations; moreover the amplitude of the perturbation vector $\Delta\Gamma$ is gradually reduced. With a proper choice of the number of iterations and the $\Delta\Gamma$ decay law, we have always found convergence for all the propellers designed.

Figure 1 shows a plot of some sample circulation distributions obtained during a run of the Optimisation Routine. It must be observed that we started form an unlikely rectangular circulation distribution, and after some initially crude tentatives, the program converged to a very smooth circulation distribution. In Figure 2 the corresponding growth of the efficiency versus the computer CPU time (on a Personal Computer with Pentium processor) is shown. It can be observed that the maximum efficiency is practically reached after 30 seconds of calculation, the remaining time being just necessary to smooth the blade surface. The grid used was M=30 and N=20.

In Figure 3 another significant feature of the method is shown. Two optimum circulation distributions, obtained for the same design conditions, are presented; one has been calculated with the correct viscous resistance and the other one setting the viscous resistance to zero. In this test case the input diameter (Dmax) was greater then the optimum one (Dopt); then in the viscous calculation the code, during its trial and error procedure, recognised that some outward sections had to be discarded due to their excessive viscous resistance, supplying at the end the value of the optimum diameter.

It is clear that this procedure of optimisation allows the addition of several different constraints in a easy way from the computational point of view; for example we can add the constraint of a minimum thickness of the blade, useful on small propellers to design realistic sections at the tip, or we can decide to discard those propellers that have overlapping blades. In this case the result will be the optimum possible propeller with the prescribed minimum blade thickness and no overlapping.

In a more extensive sense the optimisation can be performed not only on the base of the pure efficiency, but also including other aspects such as that economical, the weight, the feasibility and so on, of course once we are able to give the correct weigh to all these factors.

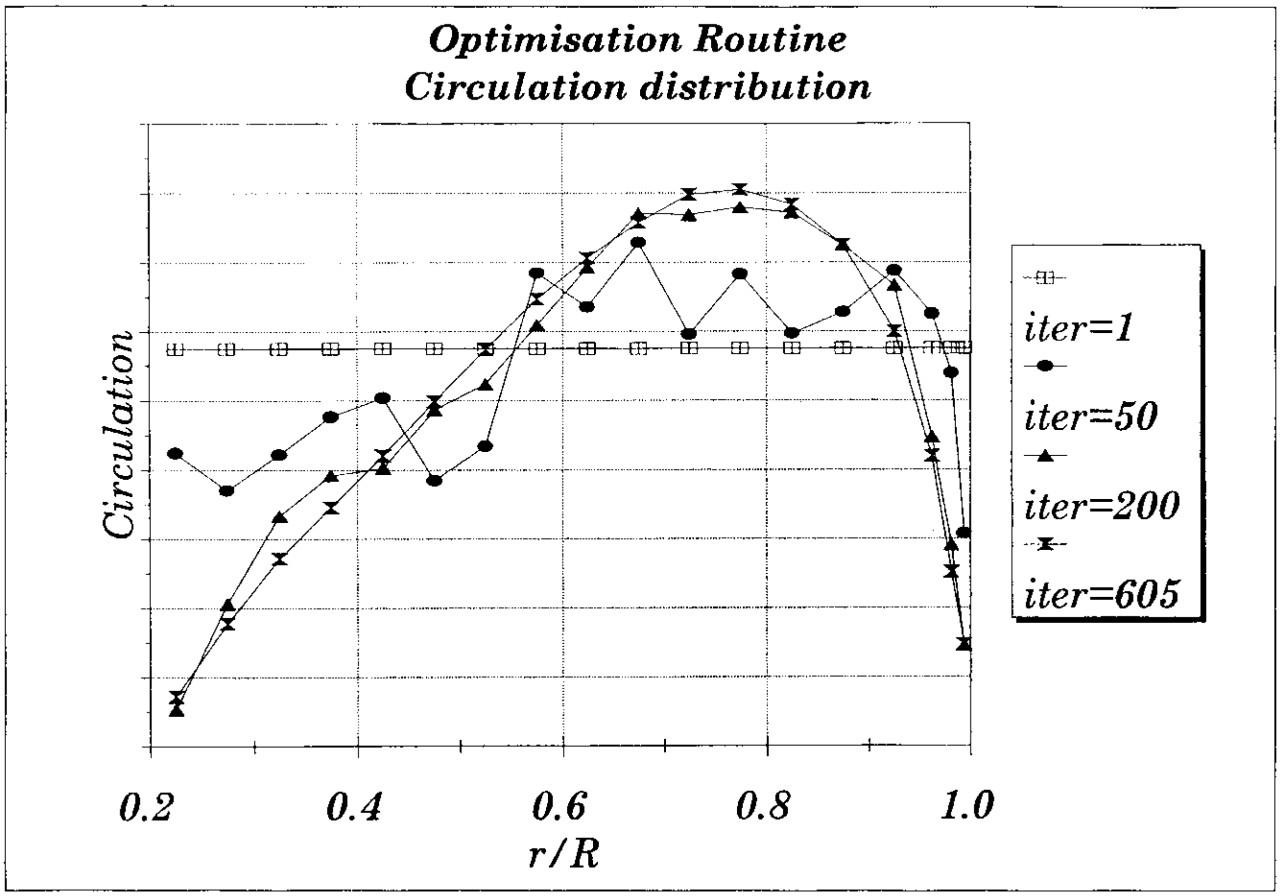

Figure 1

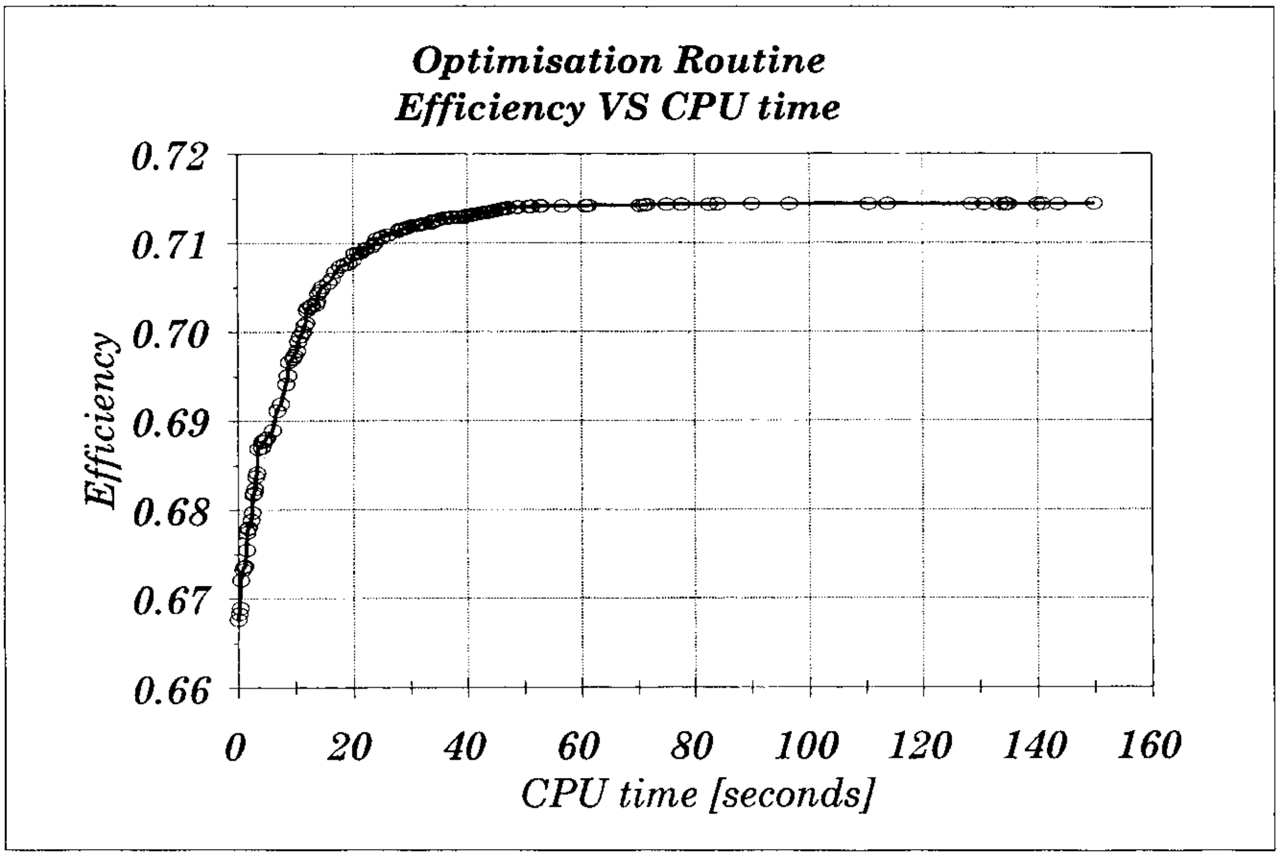

Figure 2

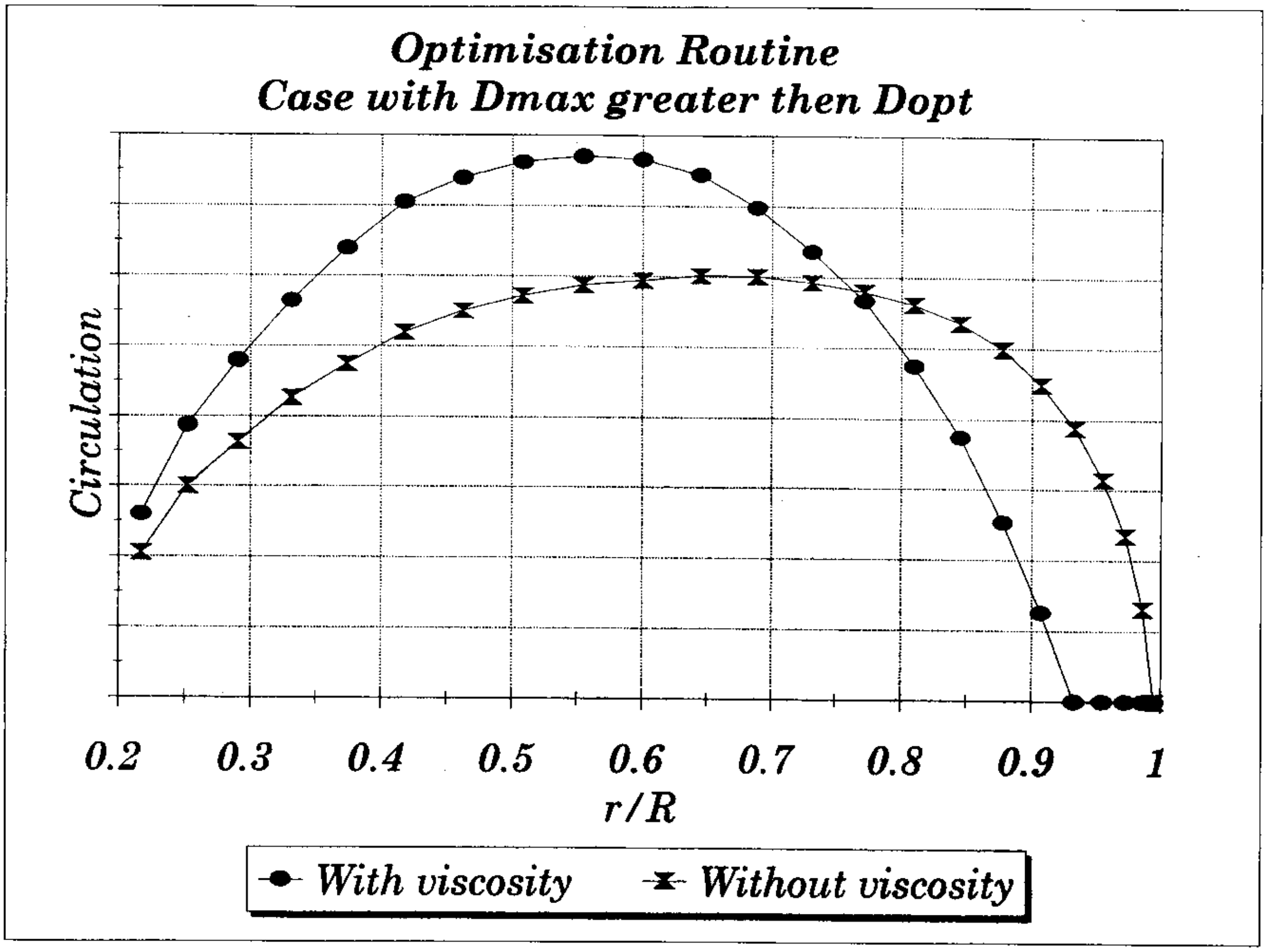

**Figure 3**

## 3 THE DESIGN ROUTINE

Once the optimum circulation, chord and thickness distribution, supplied from the Optimisation Routine output, are defined, we have to calculate the pitch and camber distributions that are really capable to generate that circulation and hence the desired thrust or torque. While there are infinite combinations of pitch and camber able to generate the given lift, if we include the additional constraint of a "shock free" entry at each section, the problem admits in general only one solution. The shock free condition states that the flow must have a stagnation point exactly at the leading edge of the profile and this carry to the avoidance of a suction peak. In general shock free propellers have the best margin to back and face sheet cavitation in non-uniform flow.

The lifting surface codes is normally used for this purpose, but we wanted to avoid some of the approximations lying in this theory, mainly due to the representation of the blades with their "mean" camber surface and to the exclusion of the hub that, as already mentioned, is sometimes inadequate for fast propellers design.

A more accurate theory, called Panel Method, has been successfully developed in the last years; since the true three dimensional shape is represented, a precise flow calculation is allowed also on thick bodies and at the blade leading edge. Normally the Panel Method is used as an analysis tool to solve the direct flow problem, namely to calculate the pressure distribution (Figure 4) once the blade and hub geometry are defined. On the contrary the inverse method, that is the calculation of the blade geometry for a prescribed pressure or load distribution, is a more difficult task from the mathematical point of view.

Once again we have skipped the problems arising from the complexity of the inverse problem, thinking to use an analysis Panel Method in an iterative way with a trial and error procedure. Starting with an arbitrary (reasonable) pitch and camber distribution, the propeller is tested with a Panel Method, checking the corresponding circulation distribution and the flow behaviour at the leading edge. From this analysis it is decided how to modify the pitch and camber for the next iteration in order to go toward the desired conditions.

In the panel code the blade, hub and wake surfaces are discretized with a great number of "panels" lying on the their exact surfaces. If N is the total number of panels lying on the solid boundary and M are the stream wise strips of panels along the wake, the potential flow problem is reduced to the following set of equations:

$$\sum_{j=1}^{N} A_{i,j}\mu_j = \sum_{j=1}^{N} B_{i,j}\sigma_j + \sum_{k=1}^{M} W_{i,k}\omega_k \qquad i = 1,\dots,N$$

**Equation (2)**

$A$, $B$ and $W$ are called matrices of influence; they depend only on the geometry of the panels and on their reciprocal position and can be easily calculated. The vector $\sigma$ is the source term; each element is given by the scalar product between the vector normal to the local panel surface and

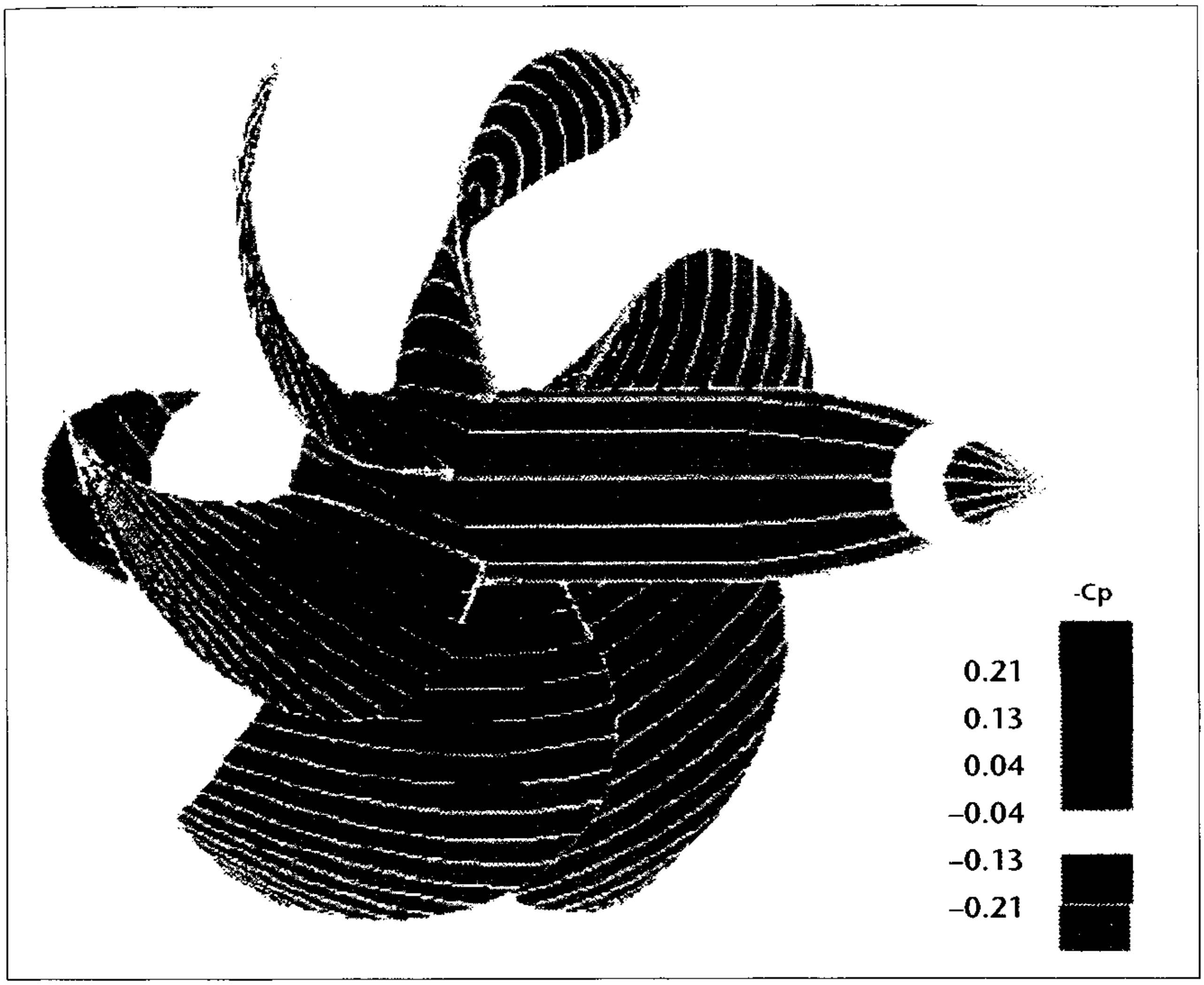

**Figure 4**

the undisturbed velocity, and hence is a known quantity. The unknowns of the problem are the N values of μ and the M values of ω. The former is the potential of disturbance on the solid boundary, while the latter is the potential jump across the wake. Even if Equations (2) do not form a set of linear equations, their solution is normally linearised inverting the matrix *A* and solving for μ with an assumed set of values for ω. The process is repeated with updated values of ω, using the Newton-Rhapson method, until the difference of pressure along the blade trailing edge goes to zero (Kutta condition). When the potential disturbance μ is known, it is possible to calculate the velocity and the pressure distribution over the propeller surface.

If we consider the generic blade section *i* represented in Figure 5, our problem is to find the correct combination of pitch and camber at all the M sections such that the circulation is equal to the prescribed value Γi and the flow has its stagnation point at the leading edge. It can be seen that this condition is satisfied when at all the M sections is:

$$i = 1, \ldots, M$$

$$\Delta_{te} = \omega_i - \Gamma_i = 0$$

$$\Delta_{le} = \mu_i^+ - \mu_i^- = 0$$

**Equation (3)**

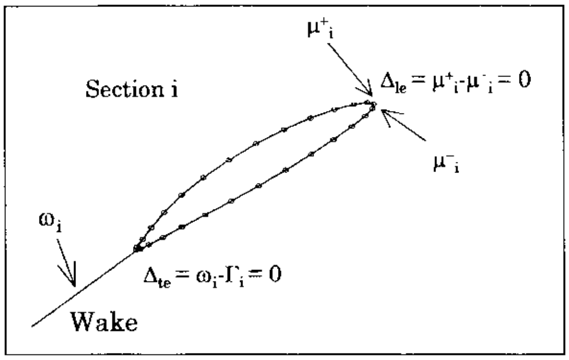

**Figure 5**

In Figure 5 $\mu_i^-$ and $\mu_i^+$ are the disturbance potential of the two panels adjoining the leading edge respectively from the pressure and the suction side. Solved Equation (2), from the analysis of the M couples of values $\Delta_{te}$ and $\Delta_{le}$, that represent the error, it is possible to calculate a better approximation of the pitch and camber distribution and to recalculate Equation (2) for the new propeller geometry. With a proper choice of the convergence technique, about twenty iterations are required to set the errors $\Delta_{te}$ and $\Delta_{le}$ to values negligible from the engineering point of view. This represents a great computational time effort, since we have to recalculate, for each new pitch and camber distribution, the corresponding matrices *A*, *B* and *W* and to invert the matrix

$A$ at each iteration. To save time, the influence matrices are kept frozen and only the vector containing the N values of $\sigma$ is updated for any new geometry to solve Equation (2). This means that Equation (2) is linearised relatively to the first geometry tested, hence committing an error that depends on how much the blade geometry used to calculate the influence coefficients differs from the actual blade geometry. Since the Design Routine is repeated several times going back and forth from the Optimisation Routine with updated blade geometry, at the end the error becomes negligible. Figure 6 shows a typical convergence history of the pitch and camber for a test propeller at section r/R=0.7, while Figure 7 represents the corresponding calculated pressure distribution at convergence, obtained imposing a NACA 66 thickness distribution and camber line NACA a=0.8. It must be also pointed out that going out from the Design Routine, at better approximation of the trailing wake is calculated, aligning the wake panels to the calculated stream lines. Moreover at each iteration the viscous effects on lift due to the boundary layer displacement are calculated with a strip wise two dimensional boundary layer analysis and the so called transpiration technique.

## 4 DESIGN SAMPLE

The application of the design code for a 52 ton fast yacht is presented. The engine power was 2X1500 hp at 2300 RPM; the full load speed had to be higher then 34 knots with low vibration levels; in order to meet the latter constraint a six bladed propeller with 35° of skew was selected.

Figure 8 shows the preliminary analysis of the open water efficiency for different reduction ratios and propeller diameters. It can be noticed the great advantage of choosing propellers with a large diameter and low revolution speed. Due to practical constraints it was selected a diameter D=1.115 m and a reduction ratio R.R.=4. In order to allocate these quite large propellers, two deep tunnels where necessary on the bottom of the hull.

Figure 9 shows the time history of the propeller efficiency versus iteration number calculated at the last run of the Optimisation Routine, that carried to a final isolated propeller efficiency above $\eta_0 = 0.77$. In the same graph are also shown the time histories of the calculated t/C, C/D and $\Gamma$ for the section at r/R=0.7. The main characteristics of the final propeller where an expanded area ratio Ae / Ao = 1.05 and an unusually high pitch ratio of $(P/D)_{r/R=0.7} = 2.1$.

At the sea trials the boat attained a maximum speed of 35.4 knots at full load. Propellers with very high P/D ratios are more efficient and work at a relatively higher cavitation number compared with conventional propellers, but they are subjected, during a revolution, to a greater variation of the angle of attack, due to the inclination of the shaft. This can lead to cavitation and damage of the blade, particularly at the face, if the design is not properly performed. Fortunately there is room enough to improve the performances of conventional NACA profiles from the point of view of face cavitation. A new class of profiles has been developed for this purpose using the Eppler code (Shen and Eppler, 1981) and the theoretical results have been confirmed with model tests at the high speed cavitation tunnel of the Ecole Politecnique de Lausanne.

Compared with conventional profiles obtained by the superposition of the NACA 66 thickness distribution and the NACA a=0.8 camber line, the new profiles have a greater leading edge radius and the maximum thickness moved forward (Figure 10). In addition more load is carried toward the trailing edge of the profile without occurrence of flow separation at the Reynolds numbers typical for these applications. Figure 11 shows, for this sample case, a comparison between the cavitation bucket of the NACA 66 and the new profile (2MC2016) designed for section at r/R=0.3.

## 6 INTERACTION BETWEEN PROPELLERS AND HULL

A final check of the propeller behaviour can be performed including the effect of the interaction between propellers and hull. An unsteady Panel Method has been developed for this purpose; since the method deals with the potential flow theory, the code is obviously appropriate only for the cases where viscous effects are negligible. For conventional fast planing crafts no flow separation usually occurs along the hull and moreover the propeller works practically entirely outside the hull boundary layer. When the hull has deep tunnels flow separation may occur for inappropriate tunnel design. Since the code, at the end of the potential flow calculation, performs a boundary layer analysis along the hull, it is possible to check the incipience of flow separation looking at the form factor of the boundary layer; if flow separation is detected the redesign of the tunnel is required.

A propeller, the hub, the trailing wake and half portion of the hull are discretised with panels (Figure 12). The propeller is set at the proper angle of inclination relative to the hull. A mirror image at the symmetry axis is used to consider the presence of the other propeller and the remaining part of the hull. Another mirror image is applied at the undisturbed free surface, hence simulating the zero Froude number condition. Moreover a wake is shed from the stern of the hull to force the Kutta condition at the dry transom.

The occurrence and the evolution of sheet cavitation is calculated using the development to the unsteady case of the method presented in Caponnetto and Brizzolara (1995).

The calculation is performed in the time domain with successive small increments of the propeller's rotation angle. Starting from rest, usually three complete revolutions are required to overcome the transitory period, and then to reach the periodicity of the flow.

The pressure is calculated at each panel and time step; its integration on the surface of the bodies and in the time supply the average forces and moments on the hull and the propellers; the comparison with the open water propeller forces are useful to evaluate the inviscid disturbance generated by the hull on the propeller, while the harmonic analysis supply the amplitude of the pressure pulses at the centroid of each panel along the hull.

Figure 13 shows, for example, the blade rate pressure amplitude on the hull above the propeller, calculated for a four bladed propeller in a tunnel, while in Figure 14 is represented, for the same test case, the boundary layer thickness along a stream line calculated with and without the presence of the propeller.

## CONCLUSIONS

A new approach for the design of marine propellers has been presented in the paper. The method utilises, in an iterative way, a Vortex Lattice and a Panel Method to calculate the geometry of an optimum propeller, taking into account viscous effects, the complete three dimensional shape of blade and the presence of the hub, within the constraints of cavitation avoidance and structural strength. Additional constraints can be added in a straightforward way.

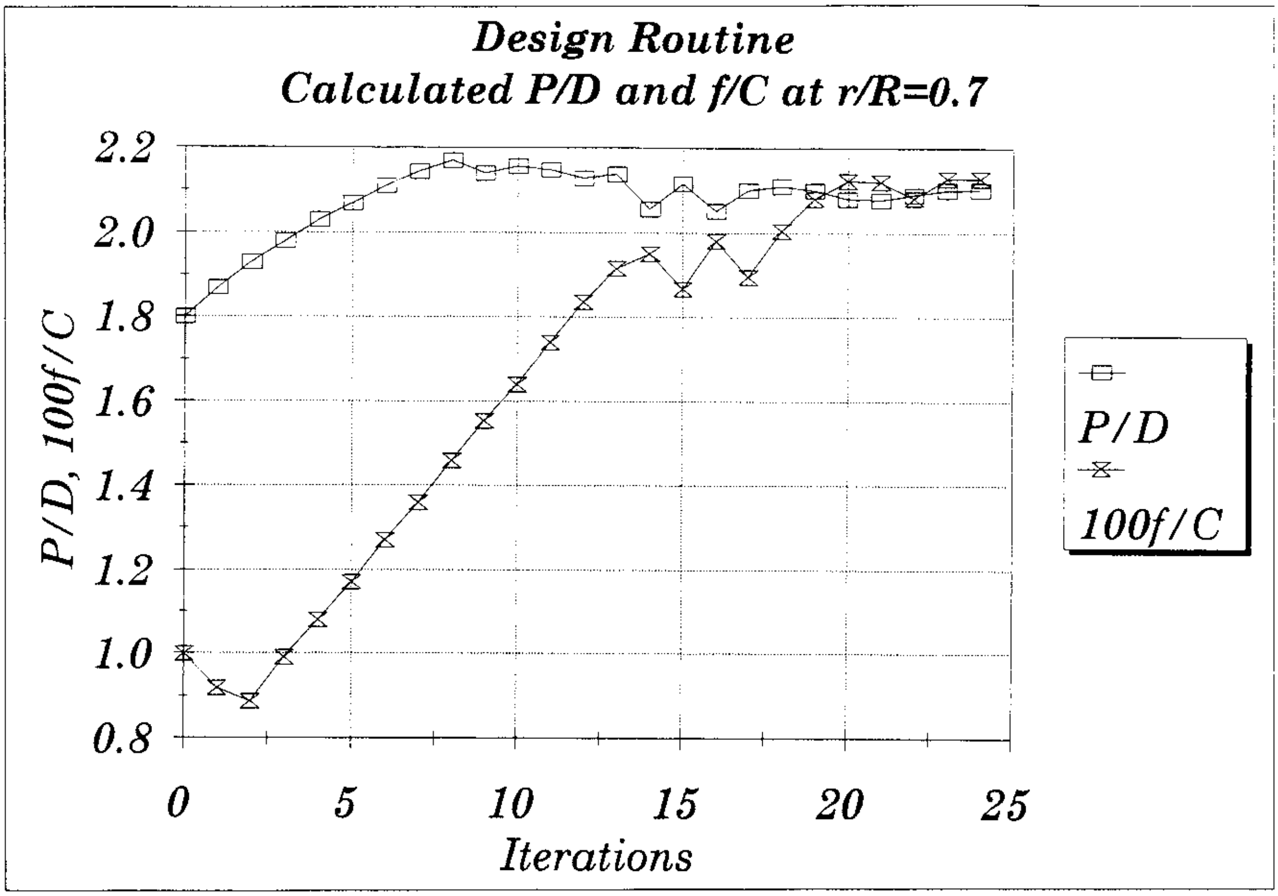

Figure 6

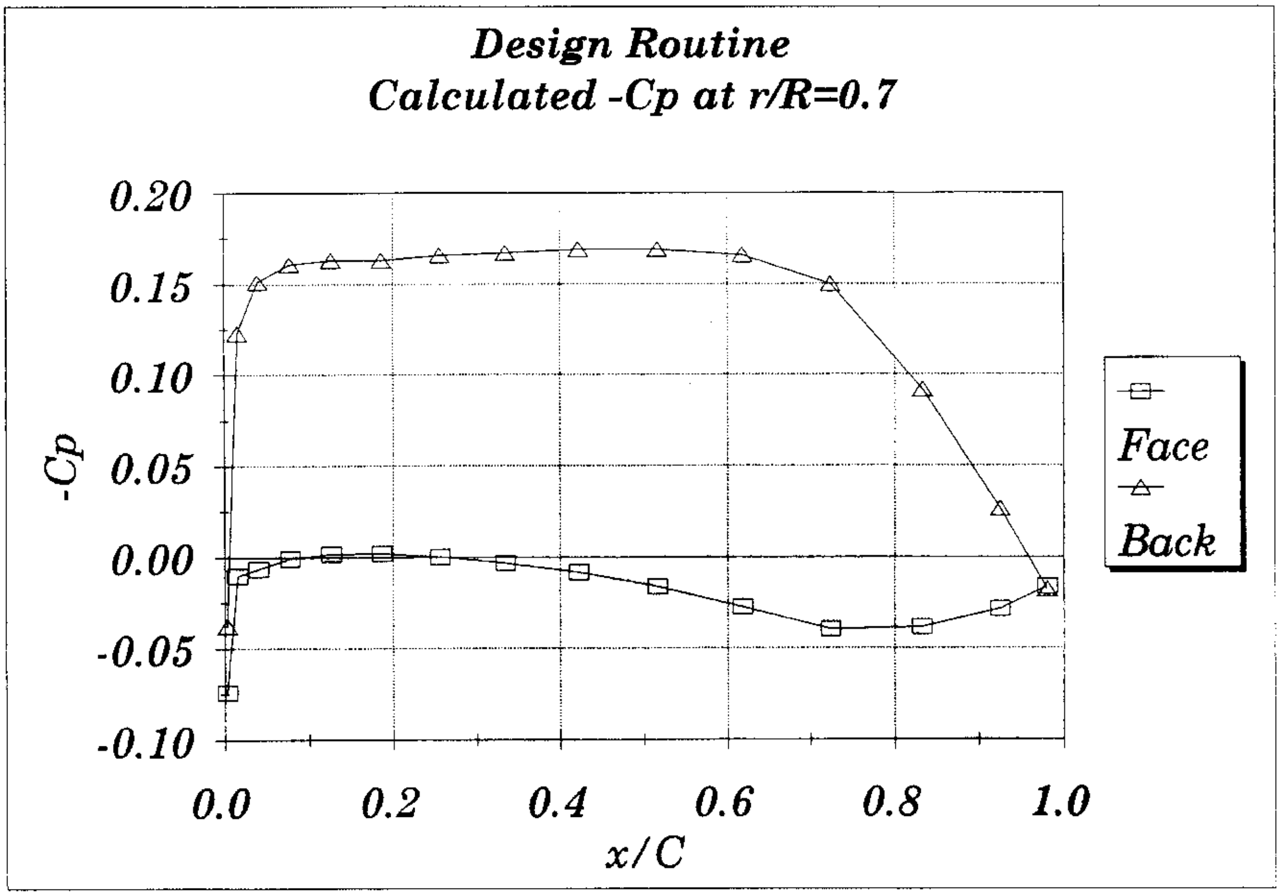

Figure 7

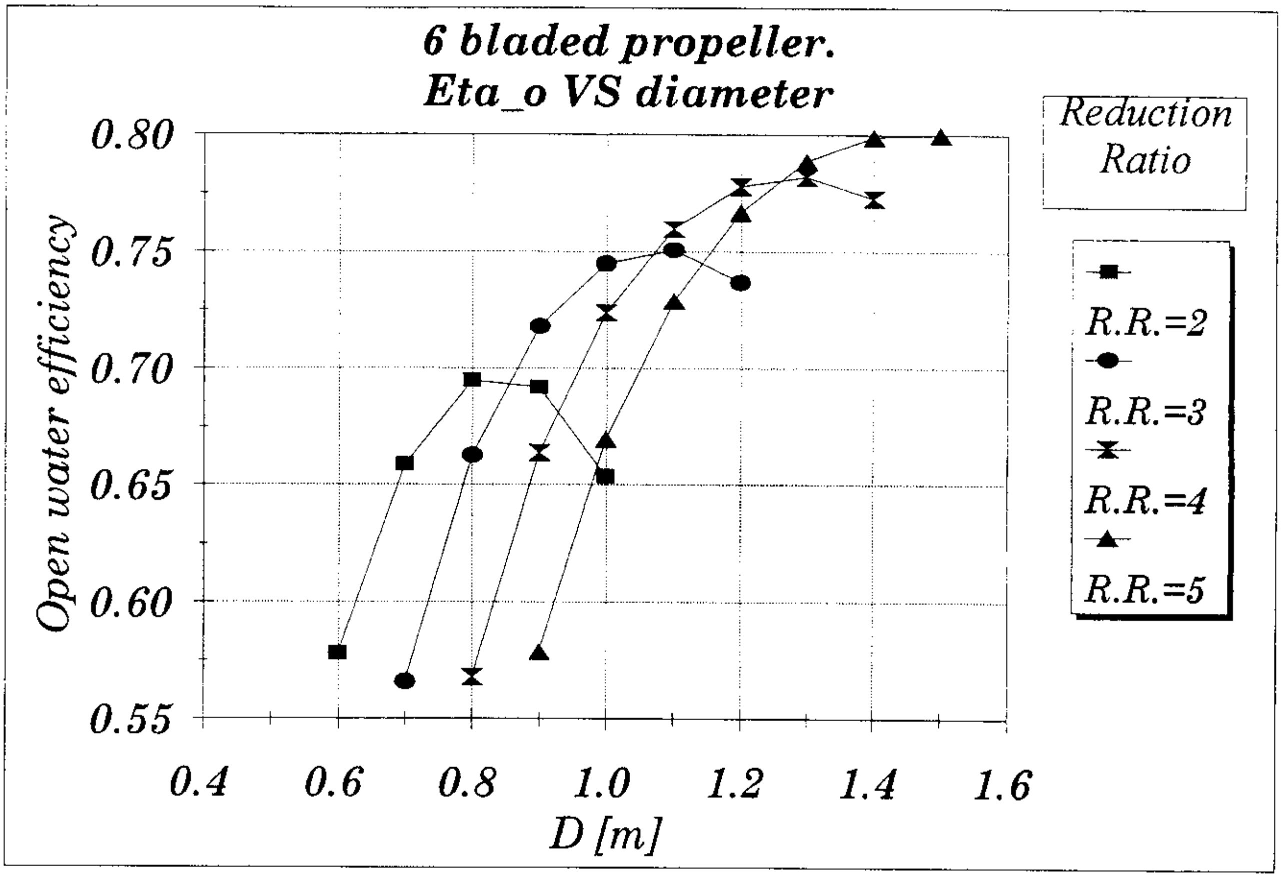

Figure 8

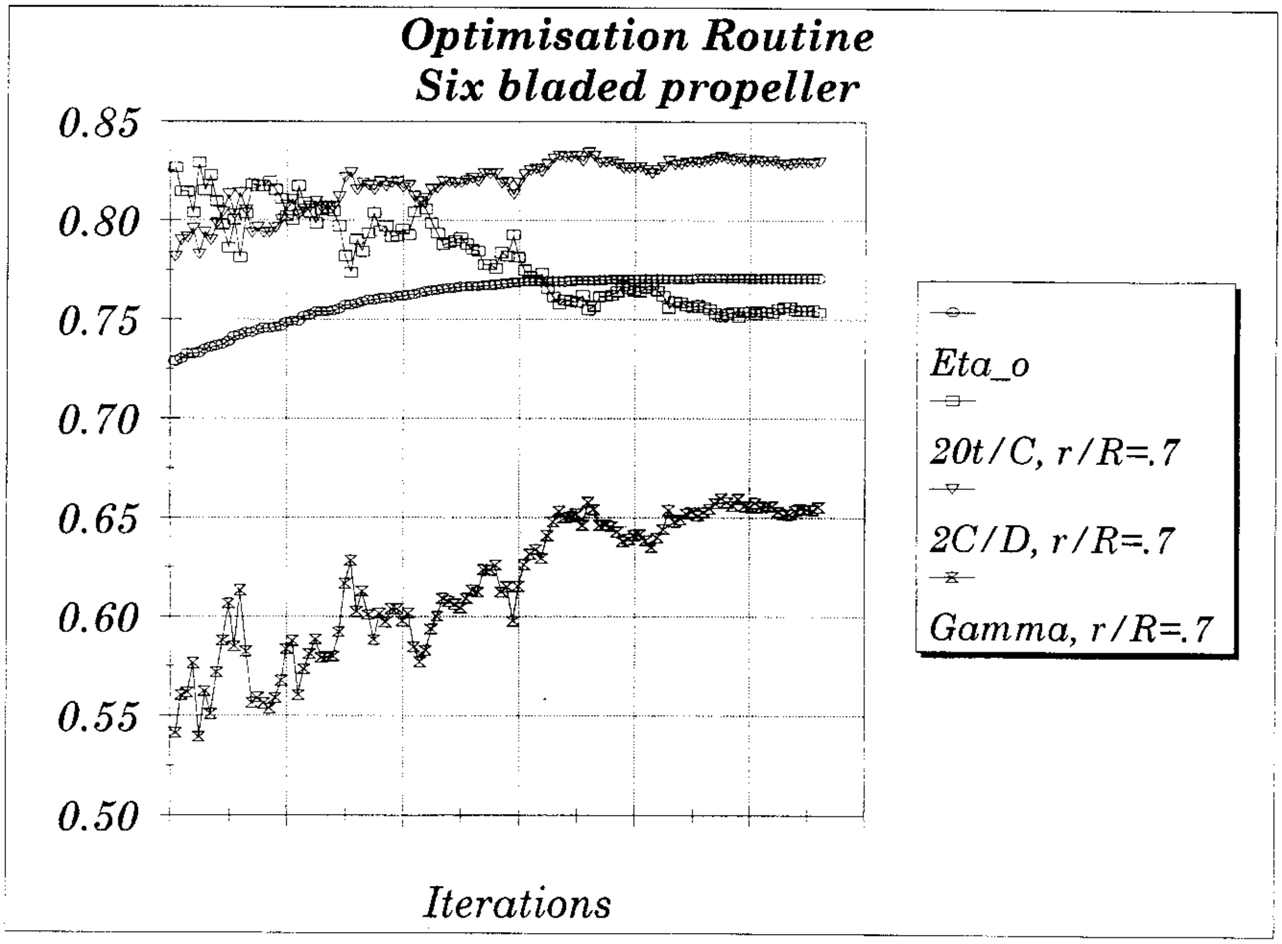

Figure 9

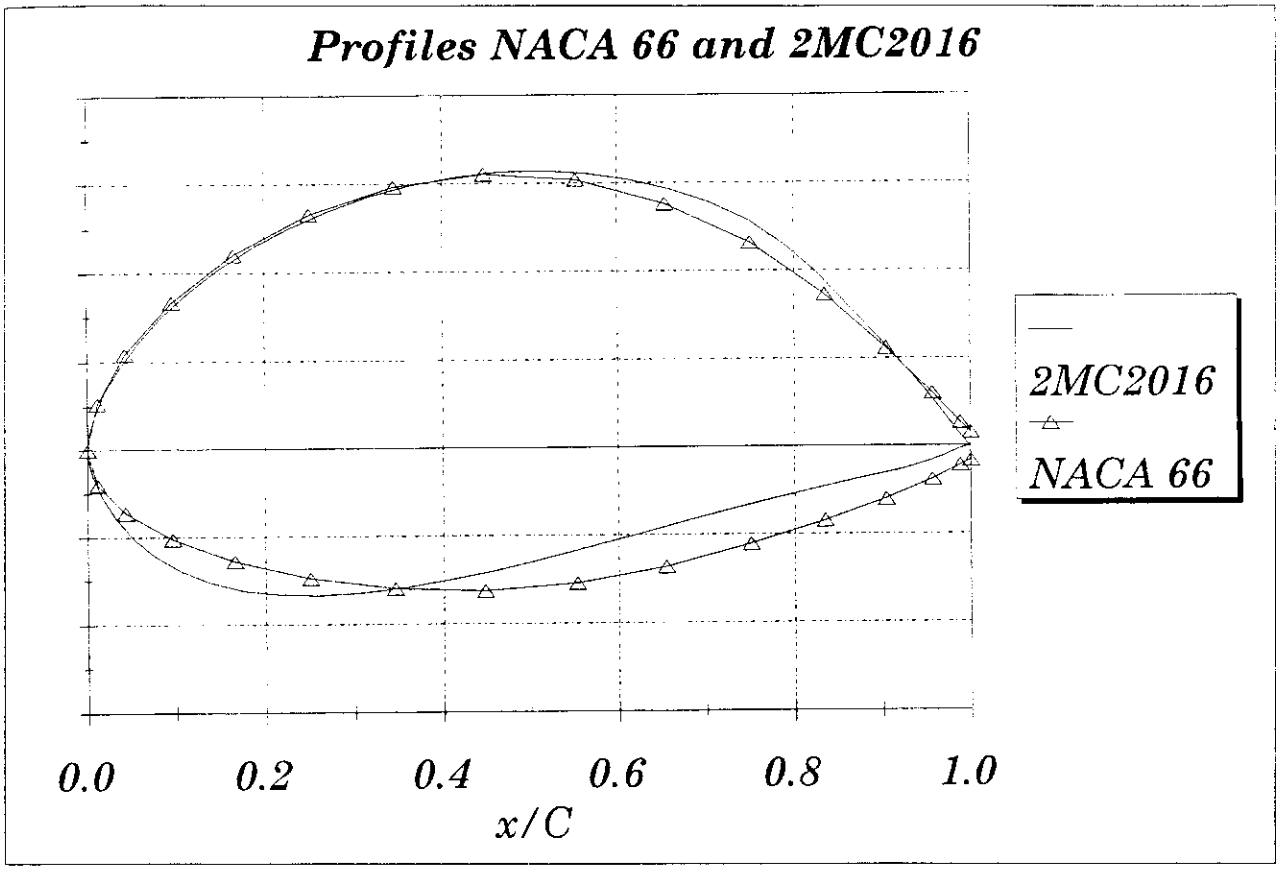

Figure 10

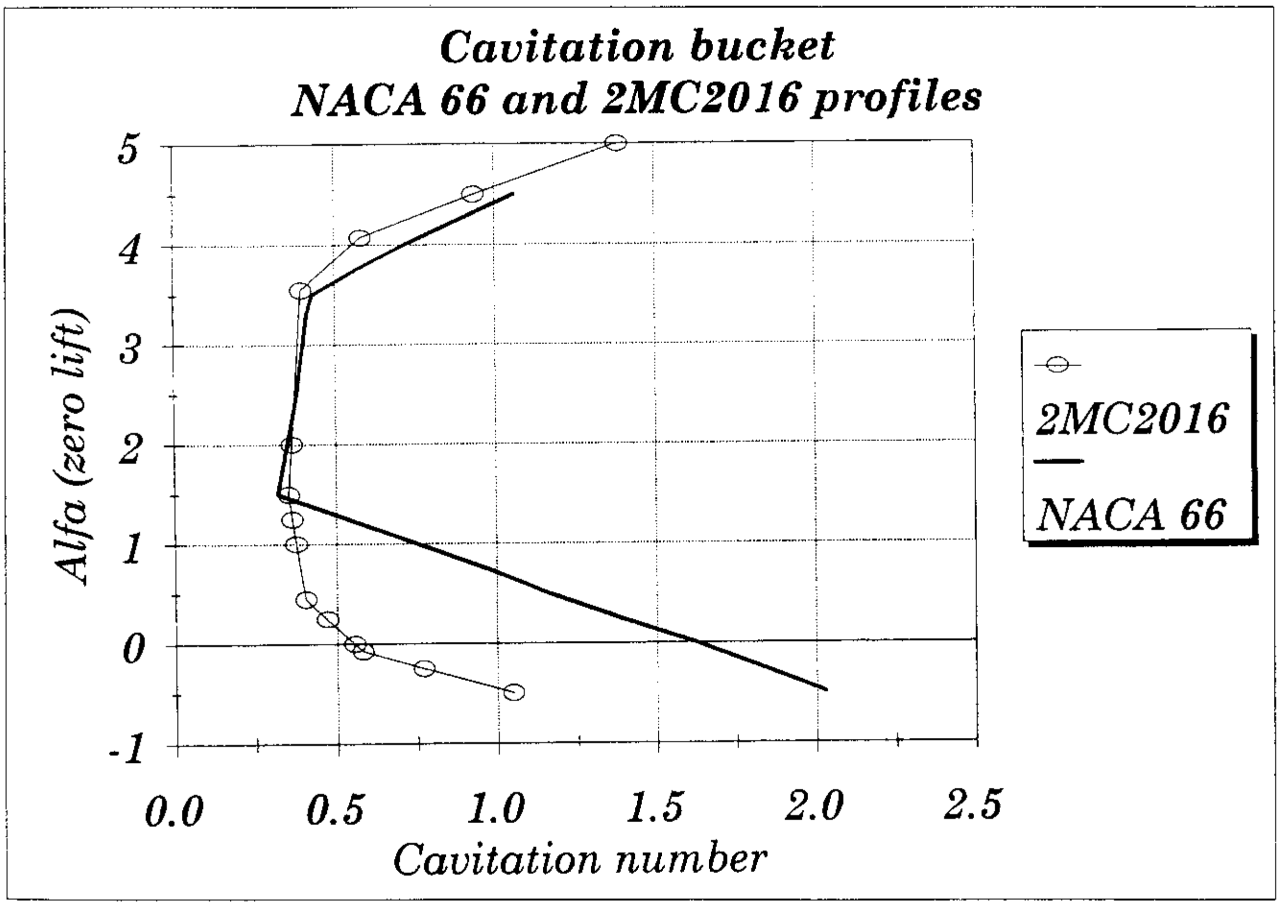

Figure 11

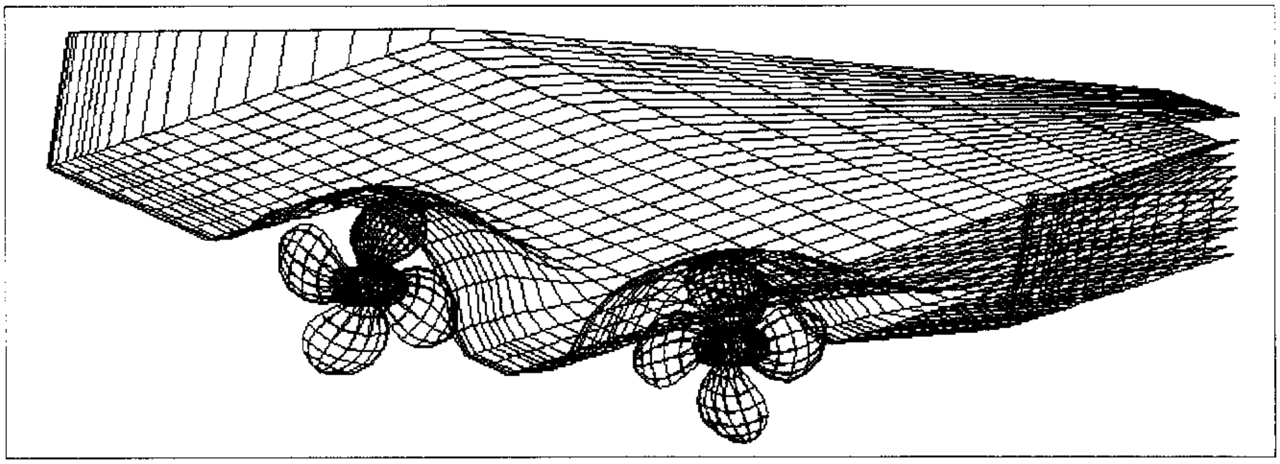

Figure 12

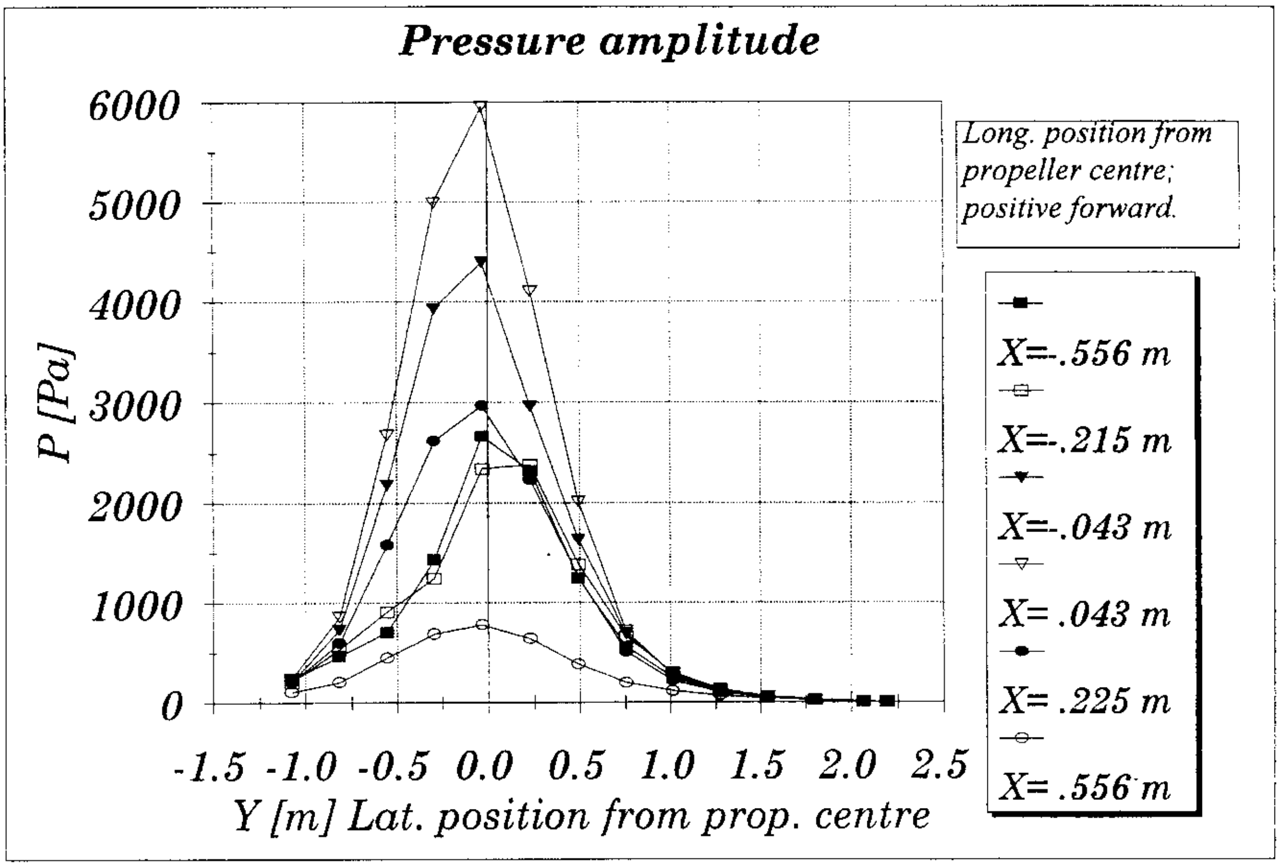

Figure 13

The application of the design procedure for a fast planing boat has been presented. The potentiality of new profiles designed to reduce cavitation on fast propellers having an inclined shaft are compared with conventional NACA profiles.

The non stationary behaviour of a propeller working near the hull of a planing boat can be calculated using an unsteady Panel Method. This code is not only useful to assess the performances of the propeller, but also to evaluate the effectiveness of the aft portion of the hull shape on planing boats having deep tunnels.

## REFERENCES

Brockett, T., 1966. "Minimum Pressure Envelopes for Modified NACA-66 Sections with a=0.8 Camber and BuShips Type I and Type II Sections", Hydromechanics Laboratory, Research and Development Report 1790. Washington D.C.: David Taylor Model Basin – Department of the Navy.

Caponnetto, M. and Caprino, G., 1994. "Application of a Surface Panel Method to Propeller Flow Analysis", NAV 94, Rome, Oct. 5–7

Caponnetto, M. and Brizzolara, S., 1995. "Theory and Experimental Validation of a Surface Panel Method for the Analysis of Cavitating Propellers in Steady Flow", PROPCAV 95, Newcastle upon Tyne, May 16–18

Greeley, D.D. and Kerwin, J.E., 1982. "Numerical Method for Propeller Design and Analysis in Steady Flow", Trans. SNAME, Vol. 90

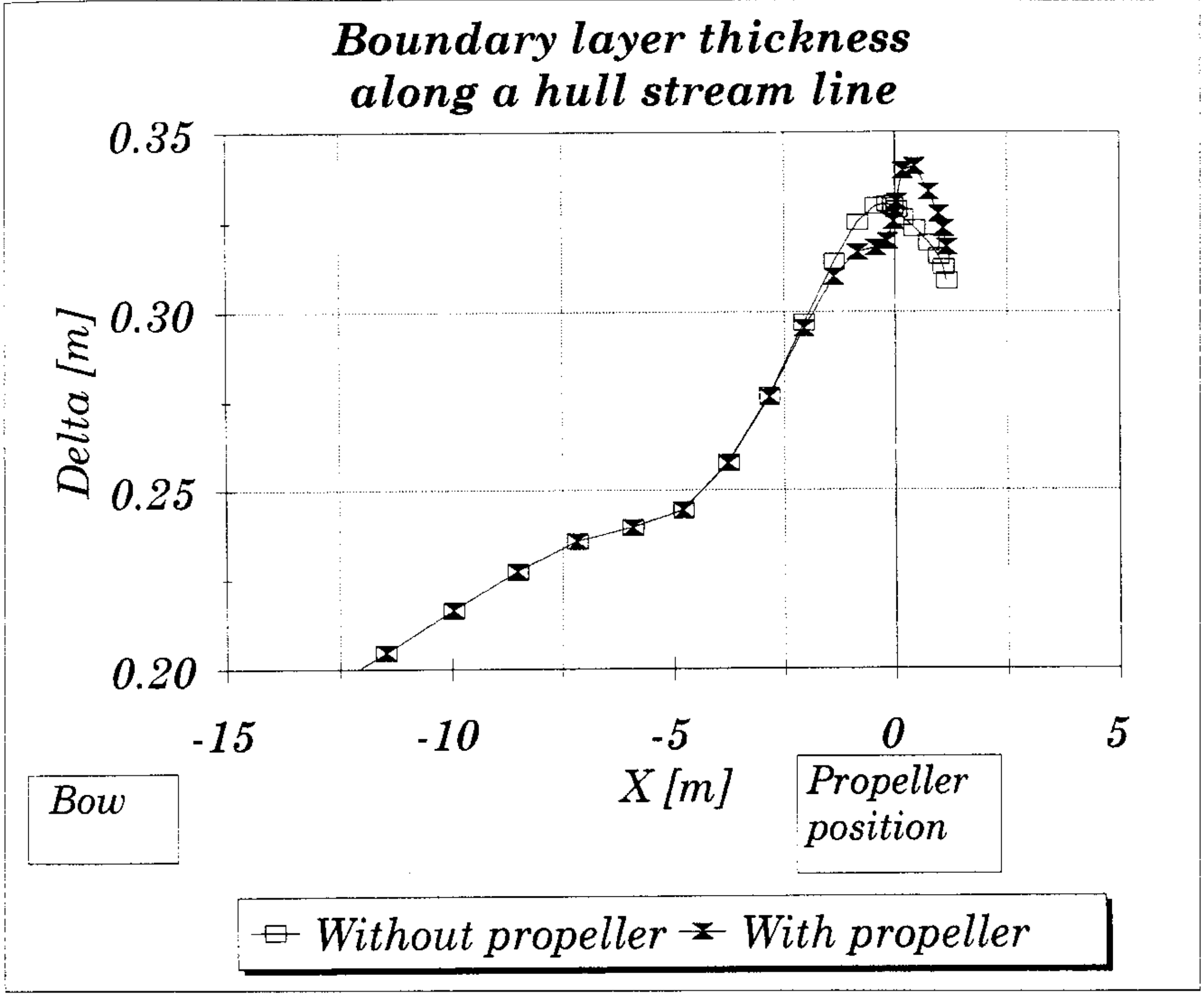

Figure 14

Hsin, C.Y., Kerwin, J.E. and Kinnas, S.A., 1991, "A Panel Method for the Analysis of the Flow Around Highly Skewed Propellers", Propellers/Shafting '91 Symposium, Sept. 17–19.

Kerwin, J.E., Coney, W.B. and Hsin, C.Y., 1986, "Optimum Circulation Distribution for Single and Multi-Component Propulsors", In Proc. of Twenty-First American Towing Tank Conference (ATTC), ed. R.F. Messalle, pp. 53–62. Washington, D.C.: National Academy Press.

Morgan, W.B., Silovic, V. and Denny, S.B., 1968, "Propeller Lifting-Surface Corrections", Trans. SNAME, Vol. 76.

Morino, L. and Kuo, C.C., 1974, "Subsonic Potential Aerodynamics for Complex Configurations: a General Theory", AIAA Journal, Vol. 12, No. 2, pp. 191–197.

Shen, Y.T. and Eppler, R., 1981, "Wing Sections for Hydrofoils – Part 2: Nonsymmetrical Profiles", JSR, Vol. 25, No. 3.

**MARIO CAPONNETTO**

Mario Caponnetto has a degree in Naval Architecture and Marine Engineering from the University of Genoa. After completing his degree in 1987, he spent two years as an officer in the Italian Navy before taking up employment in the Naval Hydrodynamics Research Office of the Fincantieri shipyard. From 1990 to 1992, Mario worked as a member of the research office of the Il Moro di Venezia America's Cup team. Since 1992 he has been a researcher at the University of Genoa where he lectures on the subject of sailing yacht design. Mario has been a consultant to Rolla SP Propellers since 1995.

**MARZIO PORRO**

Marzio Porro has been responsible for Submerged Propeller Systems at Rolla SP Propellers since 1989. Before that time he was Chief Engineer of the Mechanic and Hydraulic Systems Office at AERMACCHI Aerospace Industry SpA for eight years. Marzio holds a Degree in Aeronautical Engineering from the Polytechnic of Milan.