THE 14th CHESAPEAKE SAILING YACHT SYMPOSIUM

Sailing Yacht Design Using Advanced Numerical Flow Techniques

Mario Caponnetto, Alessandro Castelli, Philippe Dupont
Hydraulic Machines Laboratory, Ecole Polytechnique Fédérale de Lausanne, Switzerland

Bernard Bonjour, Pierre-Louis Mathey, Stephane Sanchi, Mark L. Sawley
Fluid Mechanics Laboratory, Ecole Polytechnique Fédérale de Lausanne, Switzerland

ABSTRACT

The 30th America's Cup will be held in New Zealand, commencing in October 1999. For the first time a Swiss team, the FAST2000 Challenge of the Club Nautique Morgien, will compete. Three laboratories of the EPFL (Ecole Polytechnique Fédérale de Lausanne) are collaborating with FAST2000 in the design of the boat that will race in the Cup challenges. Present-day design of IACC racing yachts relies on the use of numerical flow simulations to obtain a competitive edge. The computation of the complex hydrodynamic and aerodynamic flows around sailing yachts provides valuable information to supplement the more conventional empirical and experimental design techniques. Such flow simulations, however, are extremely challenging and thus often require state-of-the-art numerical techniques and computer technology. A number of the issues critical to IACC yacht design are discussed, and various approaches described to address them through the use of advanced numerical flow simulation.

INTRODUCTION

Since a number of years, computational methods – in particular numerical flow simulations – have been successfully applied to the design of sailing yachts (Larsson, 1990, Flay, 1996, Milgram, 1998). Even though experimentation remains the tool most commonly used by designers to obtain accurate values of the hydrodynamic and aerodynamic forces acting on the boat, numerical simulations have some major advantages. In particular, they are relatively inexpensive and fast to use, so that it is possible to test and select different candidate geometries before setting up models for the towing tank or wind tunnel. Moreover, they allow the visualisation of several quantities – such as the flow stream lines, the wave profiles or the pressure distribution – that are very difficult to obtain from experiments. This is a very useful aid for the designer to understand the physics of the flow phenomena, at least from a qualitative point of view.

Most of the numerical simulations undertaken to date in this field have been based on potential flow theory, which reduces the complexity of the Navier-Stokes equations governing the flow and, consequently, the computational resources required. In particular, a large effort has been devoted to develop reliable tools (such as the panel method) for the computation of the wave resistance, as well as the lift and drag of appendages and sails. In some cases the basic hypothesis of the theory (inviscid, irrotational flow) is approximately satisfied, however, in a number of situations it has been shown that viscosity plays a fundamental role that can not be neglected. Nowadays, computational resources exist that allow the numerical simulation of complex viscous flows around three-dimensional bodies, thus improving the accuracy of the computed flow solution. While, to date, the computational-intensive nature of such simulations has severely limited their application to sailing boat design, these problems have been alleviated in the present study through the use of high-performance parallel computing systems.

Three general areas have been considered in the present study: wave generation at the free water surface, hydrodynamic flow around the hull and appendages, and aerodynamic flow around the sails, rig and exposed hull.

While the wave pattern and resistance generated by the boat are often computed using a potential flow code, such methods are unfortunately not suitable in the stern region where flow separation can be important. To study the influence of such effects in more detail, a Navier-Stokes solver using a Volume of Fluid (VOF) approach has been employed.

Navier-Stokes computations have also been used to predict the lift and drag on an IACC hull and appendages. Special attention has been paid to the computation and visualisation of local effects, such as vortex generation at the intersection of the fin-keel, hull and bulb.

Potential flow methods are regularly employed to calculate the lift and induced drag of upwind sails. The separated flow on the mainsail due to the mast, and the influence of the separated flow around the exposed hull section, however, can not be simulated with such methods. While such viscous flow effects on upwind sails have been computed using the Navier-Stokes solver, the greatest potential for advanced numerical simulation methods lies in the analysis of downwind sails, such as a spinnaker or gennaker.

Each of the above topics will be discussed in detail in the following sections.

HYDRODYNAMIC FORCES

Wave Resistance

The computation of wave resistance has been performed using SHIPFLOW (SHIPFLOW, 1997), a commercial panel method developed by Flowtech International AB. The main advantage of this code is the possibility to perform a non-linear computation of the wave profile, taking simultaneously into account the changes of sink and trim caused by the waves. These capabilities are of substantial benefit for sailing boat design, since the dynamic length is strongly dependent on the shape of the overhangs and the local height of the stern wave. Linear codes, where the panels are placed on the undisturbed free surface, while sufficiently accurate for the computation of wave resistance on conventional large ships, can provide inaccurate results for sailing boats.

Parabolic panels with linear source distribution, instead of flat panels with a constant source distribution, have been used for all the computations presented in this paper. This choice increases the computational time, but the results are less sensitive to the number of panels and their distribution along the hull and on the free surface. For normal hull shapes, convergence is obtained after about 10 iterations, although some oscillatory behaviour exists for boats having a very large and flat stern when tested in an upright condition. A Silicon Graphics Octane workstation has been used for these computations.

Comparison with towing tank results has shown that the wave resistance computed by the panel code is usually under-predicted. However, it should be noted that the extraction of wave resistance from the experimental residuary resistance is also not straightforward. While it is difficult to achieve realistic values for the drag at low Froude numbers, the results in the range of boat speed of 8–10 knots are generally satisfactory. The trends between different hull shapes are always well captured by the numerical simulations. Many different hulls have been computed to select the models to be tested in the towing tank, changing in a systematic manner the principle geometrical parameters. In particular, a systematic change of the stern shape has been performed in an attempt to achieve the best compromise between the upright (downwind) and heeled (upwind) performances of the boat. As an example, Fig. 1 shows the experimental and computed resistance for two boats having the same rated length but different widths at the stern, tested at fixed speed but varying the heel angle from 0 to 30 degrees.

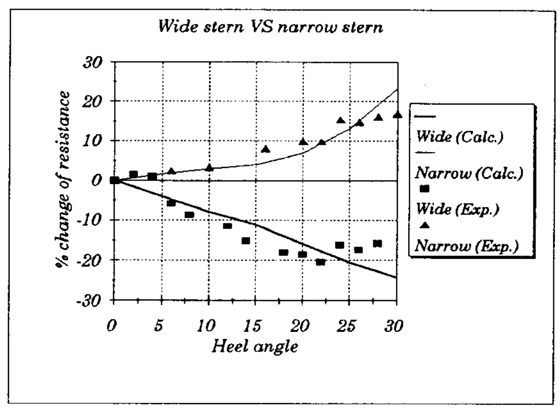

Fig. 1 Experimental and calculated changes of resistance for different stern width

In addition to the value of wave resistance, much additional information can be obtained from the numerical simulations. Naval architects usually wish to examine the wave pattern to understand the behaviour of the flow along the hull; this can readily be provided by such numerical codes.

The dynamic wetted length and wetted surface are well predicted numerically, as is the friction drag computed within SHIPFLOW using thin boundary layer theory and the pressure distribution calculated by the panel method. This information is useful in the analysis of the towing tank results, since it supplies a value for the friction drag that is more precise than that obtained using the classical ITTC methodology.

Lifting bodies can be included in the computation, but SHIPFLOW lacks the possibility to align properly the wakes. In practice, the user imposes the direction of the wake. For sailing boat design this is a drawback, since an incorrect choice of the wake path strongly affects the repartition of the lift between the keel and the rudder, and hence the total induced resistance.

Panel methods are fast, robust and can provide a great deal of useful information to the designer. While some attached viscous flow phenomena can be numerically simulated through the above-mentioned coupling with a boundary layer method, nevertheless such techniques have their limitations. The incorporation of viscous effects on the generated waves is thus best investigated through the use of a Navier-Stokes solver. Nowadays, there exist a number of different codes that are able to solve the Navier-Stokes equations with a free surface. A number of these use a deformable mesh that adjusts iteratively to the free surface (Farmer, 1995). While results are generally satisfactory, with a moveable mesh it is difficult to compute large deformations of the free surface or a multi-valued wave surface, as occurs at high speed when wave breaking occurs at the bow. For this purpose, the Volume of Fluid method provides a better approach.

In the VOF method, it is necessary to mesh both a volume of water and a portion of the air above the free surface. Starting from a flat free surface and accelerating the boat, cells at the air/water interface are filled or emptied of water, associated with the movement of the hull. Application of the VOF method to 2D flow cases – such as the computation of the impact of a wedge on a free surface (Mazufereija, 1998) – has shown that by using a very fine mesh it is possible to predict correctly large deformations of the free surface, including the formation of droplets. The VOF method can also, in principle, be applied to 3D flows; nevertheless, such time-dependent Navier-Stokes simulations obviously impose a high demand on computational resources. In the present study, 3D simulations of free-surface effects using the VOF method have been performed using a mesh with about half a million cells. With this mesh the computation for one boat speed required about 20 hours on 8 processors of a Silicon Graphics Origin2000. The computed wave profile around the hull was found to be well predicted, although the large numerical dissipation associated with such a coarse mesh leads to a limited propagation of the waves behind the stern. Nevertheless, the values of wave and viscous resistance are predicted with satisfactory accuracy.

Viscous and Induced Resistance

To determine the viscous and induced resistance for a number of different hull shapes, 3D Navier-Stokes computations have been performed using the commercial code FLUENT/UNS (FLUENT/UNS, 1997). Each hull has generally been tested at different speed and trim in order to determine the complete polar of the boat. The free surface was considered either as a flat undisturbed surface or using the deformed wavy surface computed by SHIPFLOW. The trim and sink provided by SHIPFLOW were also used to set the boat in its most realistic position. When considering the fully appended boat – with fin-keel, trim-tab, bulb, winglets and rudder – the geometry can become extremely complex. It is then very difficult and time-consuming to mesh adequately the surface of the boat and the surrounding volume using a classical structured mesh, as is the convention for naval applications. In this case, unstructured mesh techniques greatly help to model the computational volume, providing good mesh quality even for very complex geometries.

In order to calculate the skin friction and the occurrence of flow separation, a very fine mesh is required in the boundary layer. To overcome this problem a hybrid computational mesh has been adopted, using unstructured tetrahedral elements in the bulk flow, and structured prismatic elements in the boundary layer. Fig. 2 shows the surface elements together with a sectional view of the layer of cells used to discretise the boundary layer at the junction between the bulb and the fin-keel.

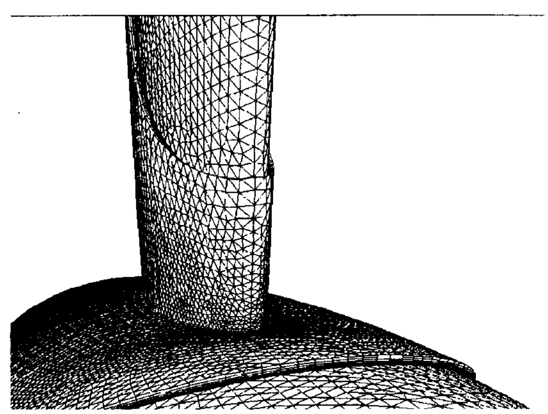

Fig. 2 Surface and boundary layer mesh at the bulb-keel junction

A total of 1.5 to 2 million cells has generally been used for the present computations, with about half of the cells located in the boundary layer. Turbulence is modelled using the two-equation RNG k-ε model. The computational time for the 500 iterations required for convergence is typically 10 hours on 12 processors of a Silicon Graphics Origin2000.

Numerical simulations of this kind are very useful to predict the effect of different hull shapes on viscous drag and flow separation. When the boat sails with a yaw angle (angle of attack), the hull and appendages develop a lift and induced drag that are determined by the vortex shedding. For wing-type surfaces, such as the keel, winglets and rudder, the location of the vortex separation can be easily calculated to be at the sharp trailing edge. More difficult to simulate numerically is the vortex that smoothly detaches from the hull; in this case the location of the detachment and the strength of the vortex is strongly dependent on the velocity distribution inside the boundary layer (Fig. 3).

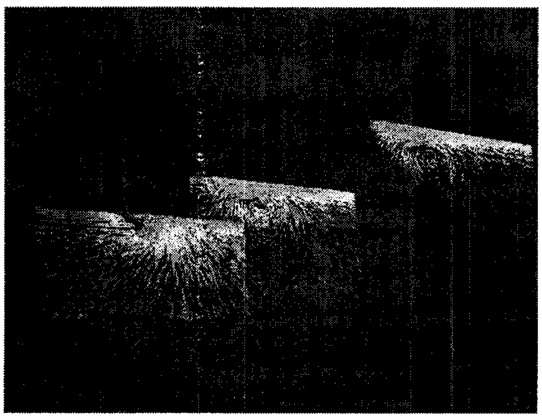

Fig. 3 Vortex separation behind the transom of an IACC boat

Additional vortices are shed at the junction between the appendages and the hull (horseshoe vortex) that generate the so-called interference drag; this effect, and the associated drag, can be reduced with properly designed fillets and strakes.

AERODYNAMIC FLOW

The computation of the aerodynamic flow around the sails of a yacht is a challenging task since it involves many different aspects of engineering. Upwind sails should function as efficient wings, developing the maximum lift with the minimum drag and heeling moment. This goal is achieved when the optimum spanwise circulation distribution is chosen and when the separation of the flow is minimised. On the contrary, when sailing downwind the aerodynamic drag on the sails contributes to the thrust, and in this case a large amount of separation around the sails is expected. It is obvious that the physics of the flow in these two cases is radically different, and different approaches must be considered for the accurate solution of each problem.

For the present study, different numerical codes have been used for these purposes. These choices reflect the two main objectives of sail computations: the calculation of precise upwind and downwind sail coefficients to feed the Velocity Prediction Program (VPP), and the determination of the optimum sail shape to aid the sail maker in the moulding or cutting of the sails.

Optimisation of Upwind Sails

To determine the sail shape that maximises the thrust for a given condition of wind speed and angle, an inverse Vortex Lattice Method has been developed. The optimisation procedure is based on a genetic algorithm, similar to the one presented in (Caponnetto, 1997) for the optimisation of propeller blades, where the optimum is obtained adding successively random disturbances to an initially arbitrary shape. With this approach several different constraints can be added very easily. Sails can be optimised with or without the constraint of maximum heeling moment. Limiting values of the sectional lift coefficient can be included to prevent stall. The viscous drag is added at each section using 2D drag coefficients that have been previously calculated using FLUENT/UNS for different combinations of camber ratio and mast dimension. During the iteration procedure, unrealistic sail shapes can be discarded. Typical is the case of mainsails having an inverse camber in the upper part; theoretically this shape maximises the thrust in strong wind conditions, but is obviously very difficult to trim. Using this code it is possible to obtain the complete polar diagram of the "best" sails in upwind conditions and their corresponding shapes.

Fig. 4 shows a sample case of how the optimum circulation distributions on the genoa and the mainsail change varying the wind speed.

In both cases a heeling moment of 30 t*m, corresponding to a heel angle of about 30 degrees, has been imposed. In medium wind conditions (Vaw=18 Kn) the main is very loaded, especially toward the head.

As the apparent wind speed increases (Vaw=30 Kn), the main must be unloaded at the tip, and in theory the circulation should be zero from the headboard to the top of the genoa. For the same sample case the corresponding twist angle and camber ratio are plotted in Fig. 5 and Fig. 6.

These results are in good agreement with the experience. In medium wind both the genoa and the main have highly cambered sections. The twist of the main is large, but the boom is close to the centreline of the boat. In stronger wind the sails must be flattened, especially the main in the upper part, to lower the centre of pressure. The head of the genoa must be still relatively fat to avoid separation. The twist of the main is reduced, but the angle of the boom is increased.

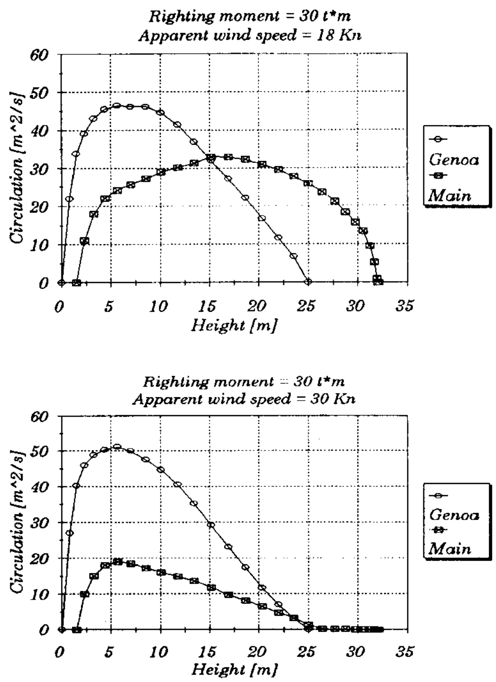

Fig. 4 Optimum circulation distribution

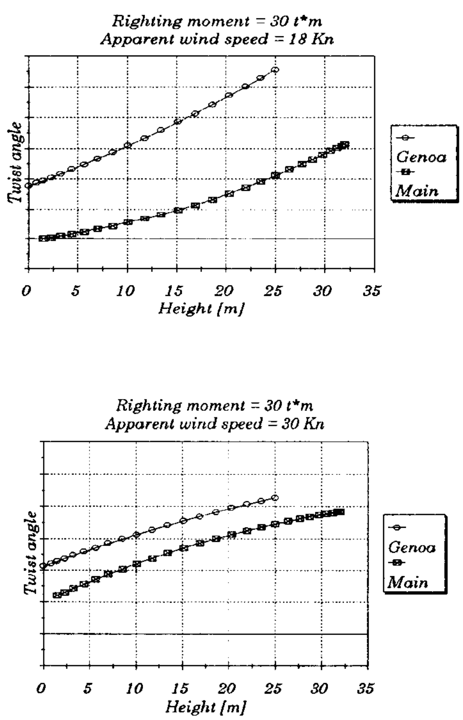

Fig. 5 Optimum twist distribution

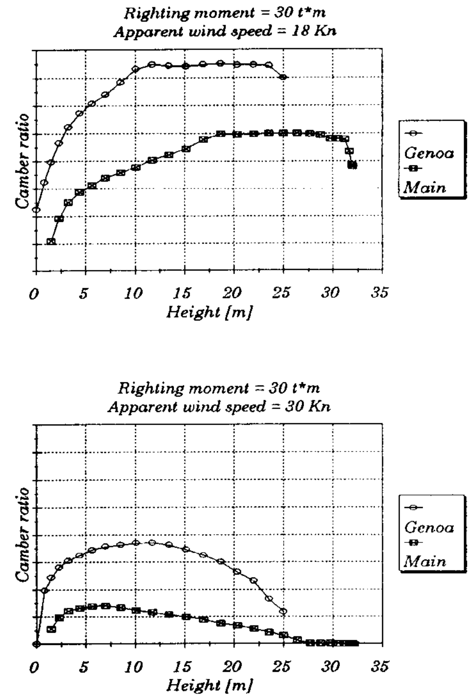

Fig.6 Optimum camber distribution

Determination of Flying Shape of Sails

Sails are not rigid wings, but thin flexible surfaces; this provides both advantages and drawbacks. An advantage is that the shape of the sail can be adjusted in the spanwise and chordwise directions by changing the tension of some wires (e.g. stays, runners) and by changing the flexion of the mast. The final shape of the sails is obtained as an equilibrium between the aerodynamic loads acting on the cloth and the internal and external loads in the cloth and the rig. The major drawback of having thin profiles is that the flow can suddenly separate if the sail is not working at the ideal angle of attack.

For the design of optimum upwind sails, it is mandatory to be able to predict the deformed shape of the sails under aerodynamic loads (the so-called flying shape). The computation of such an aeroelastic phenomenon requires a coupling between flow and structural codes. In the present study, this has been performed using the codes FLOW and MEMBRAIN, developed by North Sails Inc. FLOW employs a potential flow method for thin multiple lifting surfaces that allows an accurate relaxation of the trailing wake. The input is the initial shape of the sails (without external loads) and the result is the aerodynamic pressure distribution corresponding to a given wind speed and direction. MEMBRAIN is a structural finite element code developed to calculate the stresses and the deformations of the sailcloth and rig under the action of a prescribed aerodynamic load distribution. FLOW and MEMBRAIN are used iteratively to compute the final sail shape and rig deformation satisfying simultaneously the aerodynamic and structural equilibrium. Fig. 7 shows the results of such a coupled computation, where the initial and flying shapes of a sail plan can be compared.

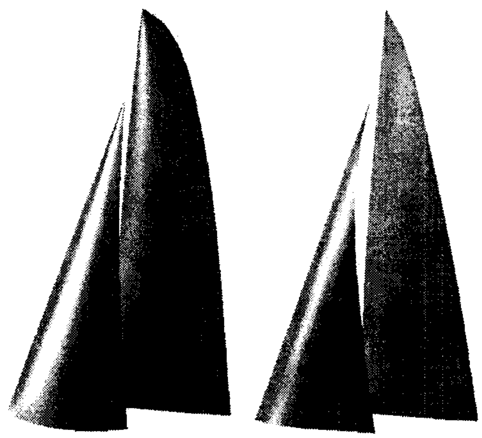

Fig. 7 Design (left) and flying (right) shapes, the latter determined by FLOW-MEMBRAIN aeroelastic computations

Even using a standard PC with a 300 MHz processor and 128 MB RAM, solving such an aeroelasticity problem with FLOW-MEMBRAIN is relatively simple and fast. It is therefore easy to compute a large number of different trims of the sails in a relatively short time (15-20 min. each).

The final aim of these numerical simulations is to find a design shape that, with the proper trim of the sheets and wires, approaches the optimum shape previously computed by the above-mentioned optimisation code.

Upwind Sail Computations

A Vortex Lattice Method supplies precise aerodynamic loads when the flow remains attached. If this hypothesis is satisfied, the lift and induced drag are satisfactory predicted. However, even well trimmed sails exhibit some amount of separation and this affects the total efficiency of the sails. Such effects can only be determined by a viscous flow computation. This problem has been addressed by using a Navier-Stokes solver, FLUENT/UNS, with the input flying shape computed by FLOW-MEMBRAIN.

Since one of the main sources of aerodynamic drag is the mast, it should be included in the computation to achieve precise pressure distributions on the mainsail. Separation behind the mast can be correctly computed only if the behaviour of the boundary layer is properly predicted.

In our computation about 60,000 triangular elements were used to mesh the surface of the mast, with the boundary layer captured using 10 layers of cells, the first cell having a height of 0.5 mm. Three million cells were used for the total computational volume, 600,000 of these cells required just to mesh the boundary layer of the mast. A computational time of about 5 hours on 64 processors of the Silicon Graphics Origin2000 was required. A number of different mast profiles have been studied using FLUENT/UNS. Fig. 8 shows both the surface mesh for a standard mast and mainsail and the computed tip vortex.

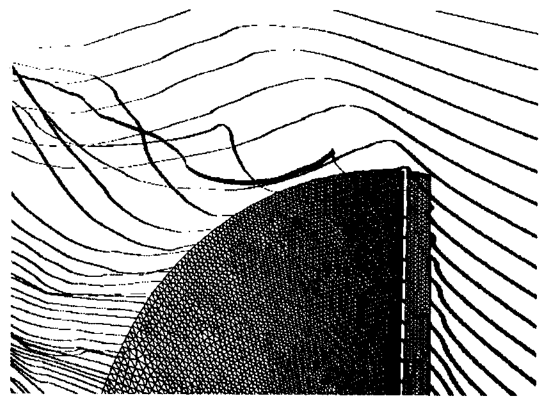

Fig. 8 Surface mesh and tip vortex path lines for a mast and mainsail configuration

For code validation purposes, a comparison between our computations and wind tunnel data (Wilkinson, 1984) for a series of 2D sail configurations has been undertaken. Fig. 9 shows that good agreement between the calculated and measured pressure distribution on a mainsail and the circular mast is obtained.

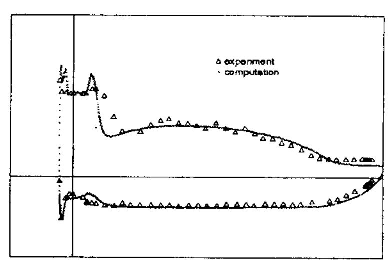

Fig. 9 Pressure distribution around a mast and mainsail geometry

Downwind Sail Computations

Due to the presence of large regions of flow separation, potential flow methods can not be used to compute the flow around downwind sails (spinnaker or gennaker). This fact, together with the difficulty in performing accurate experimental studies, has resulted in a lack of data and knowledge in the literature about this subject. A large potential for design improvement therefore exists in the use of advanced viscous flow codes. In the present study, computations have been performed using FLUENT/UNS of the flow around such downwind configurations, composed of a spinnaker (or gennaker) and mainsail, as well as the exposed portion of the hull. In addition to the determination of sail lift and drag, which can be used to feed the VPP with downwind sail coefficients, flow visualisation is very useful to understand the physics of the complex underlying phenomena.

As shown in Fig. 10, the results of such numerical flow simulations show a large separated wake region downwind of the spinnaker, comprised of a complex series of vortices.

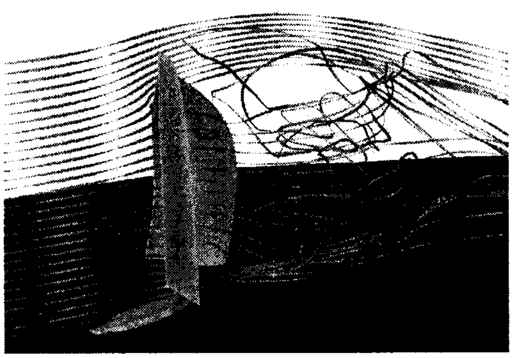

Fig. 10 Streamlines for flow around downwind sails, computed by FLUENT/UNS

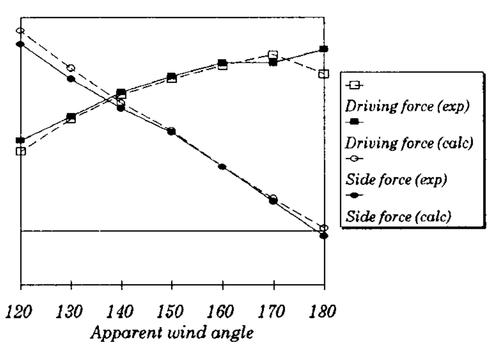

Fig 11 Comparison between calculated and experimental results for downwind sail

As expected, the flow is generally unsteady and, during the iteration procedure, the computed forces oscillate around a mean value.

Despite the complexity of the flow and the danger inherent in using a steady method to compute an unsteady flow, comparison with wind tunnel data is encouraging. This can be observed in Fig. 11, for a typical mainsail-spinnaker configuration.

A systematic study of different spinnaker and gennaker shapes, each one tested for a range of wind angle and sail trim (combinations of mainsail boom and spinnaker pole angles), has been performed, resulting in the shape and trim that maximises the thrust for each wind condition.

Fig. 12 shows a vector plot of a main and spinnaker configuration at an horizontal cut located 20 meter above the free surface.

Fig. 12 Velocity vectors for properly trimmed sails

This figure is just a two-dimensional plot of a complete three-dimensional computation. Both main and spinnaker have been trimmed to develop the highest driving force. It can be noted that the flow on the main is separated, while it remains attached on the greater part of the spinnaker. Fig. 13 shows a similar plot, but the sails are not trimmed at best. Now the flow on the spinnaker is completely separated. As a consequence, the reduction of driving force is about 40%.

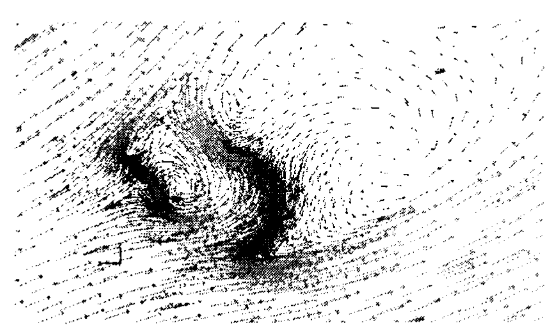

Fig. 13 Velocity vectors for badly trimmed sails

CONCLUSIONS

The present study has presented a number of different approaches that have been undertaken for the numerical simulation of the flow around a sailing yacht. It has been shown that, despite the extremely complex flow behaviour present, information can be gained that is invaluable for design purposes. This information can be either the quantitative evaluation of the hydrodynamic and aerodynamics forces exerted on the yacht, or detailed qualitative insights into the flow behaviour.

It can be recognised that the design of IACC yachts is entering a new phase through the use of state-of-the-art numerical flow solvers on high-performance computer systems. These advanced tools provide valuable replacements for previously employed empirical data and educated guessing. As a supplement to existing panel methods, they provide the naval architect with an enhanced probability of achieving a competitive design.

ACKNOWLEDGEMENTS

The present study was undertaken in collaboration with the FAST2000 Design Team. The authors wish to acknowledge the interest of the design team members (Philippe Briand, Sebastien Schmidt and Peter Van Oossanen), with whom numerous discussions have led to a greater understanding of the challenges involved in sailing yacht analysis and design. Assistance provided by North Sails Inc. and Silicon Graphics Inc. is also gratefully acknowledged. This work has received financial support from the Commission pour la Technologie et l'Innovation.

REFERENCES

Caponnetto, M., Rolla, P., Porro, M., "A New Propeller Design Method for Fast Planing Hull Applications", FAST 97, Sydney, 21-23 July 1997.

Farmer, J., Martinelli, L., Jameson, A., and Cowles, G.C., "Fully-nonlinear CFD techniques for ship performance and design", AIAA Paper 95-1690, presented at the 12th AIAA Computational Fluid Dynamics Conference, San Diego CA, June 1995.

Flay, R.G.J., "Sail aerodynamics", Journal of Wind Engineering and Industrial Aerodynamics, **63**, 1-193 (1996).

FLUENT/UNS User's Manual (Version 4.4), Fluent Inc. (1996); see also www.fluent.com

Larsson, L., "Scientific Methods in Yacht Design", Annual Review of Fluid Mechanics, **22**, 349-385 (1990).

Mazufereija, S., Peric, M., Sames, P., Shellin, T., "A Two-fluid Navier-Stokes Solver to Simulate Water Entry", Twenty-second Symposium on Naval Hydrodynamics, August 9-14, 1998, Washington D.C.

Milgram, J.H., "Fluid Mechanics for Sailing Vessel Design", Annual Review of Fluid Mechanics, **30**, 613-653 (1998).

SHIPFLOW User's Manual (Version 2.4), Flowtech International AB (1997).

Wilkinson, S., "Partially Separated Flow Around 2D Masts and Sails", Ph.D. Thesis, Ship Science Department, University of Southampton (1984).