AMERICA'S CUP YACHT DESIGN USING ADVANCED NUMERICAL FLOW SIMULATIONS

MARIO CAPONNETTO, ALESSANDRO CASTELLI, EPFL–DGM – HYDRAULIC MACHINES LABORATORY
BERNARD BONJOUR, PIERRE-LOUIS MATHEY, STEPHANE SANCHI, MARK L. SAWLEY,
EPFL–DGM – FLUID MECHANICS LABORATORY

La conception d'un voilier moderne de type 'Class America' s'appuie sur l'utilisation de simulations numériques de l'écoulement afin d'obtenir des prestations compétitives. Le calcul des écoulements hydrodynamique et aérodynamique autour de voiliers est, de toute manière, extremement exigeant; il requiert la technologie informatique et les techniques numériques les plus avangardistes. Un certain nombre de problèmes, critiques à la conception de voiliers pour l'America's Cup, sont discutés, et diverses approches liées à la simulation numérique avancée d'écoulements sont décrites.

The design of modern America's Cup 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 is, however, extremely challenging, requiring state-of-the-art numerical techniques and computer technology. A number of the issues critical to America's Cup yacht design are discussed, and various approaches described to address them through advanced numerical flow simulation.

INTRODUCTION

The 30th America's Cup will be held in Auckland, New Zealand, commencing in October 1999 with the final races scheduled for February – March 2000. For the first time, there will be a Swiss competitor, the FAST2000 Challenge of the Club Nautique Morgien [1]. Participation in this prestigious sailing competition will pit local knowledge and skills against the world's sailing elite, and thus provide the potential to display local sailing experience and know-how to a world-wide forum.

FAST2000 has embarked on a two-yacht campaign, with the first yacht being launched at Pully on 27 August 1998. This yacht, which has been designed using the knowledge and techniques that existed during the last America's Cup in 1995, will be used for crew training. Three laboratories of the EPFL (LMF, LMH and LTC) are collaborating with FAST2000 in the design of the second yacht that will race in the Cup challenges [1].

The use of advanced techniques has become essential in the design of an International America's Cup Class (IACC) yacht for the America's Cup. Since a number of years, computational methods – in particular, numerical flow simulations – have been successfully applied to the design of sailing boats. Even though experimentation remains the tool most commonly used by designers to obtain accurate values of the hydrodynamic and aerodynamics 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 streamlines, 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.

It is important to note that for IACC yachts, typical differences of speed between the winning and losing boats are about 1 – 2%. Under such conditions, it is clear that a high level of precision is required to predict boat performance to sufficient accuracy. Such precision places strong demands on both experimental and numerical methods used to determine the forces acting on the boat.

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 (keel and rudder) and sails. In some cases the basic hypothesis of the theory (inviscid flow) is satisfactory fulfilled, 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 the complex viscous flow around three-dimensional bodies, thus improving the accuracy of the computed flow solutions. While, to date, the computation-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 a high-performance computer system (Silicon Graphics Origin2000) installed at the EPFL.

The present paper briefly outlines some of the numerical simulation studies that are being undertaken at the EPFL within the framework of the collaboration with FAST2000.

Sailing Yacht Dynamics and Velocity Prediction

The forces acting on a sailing boat are the aerodynamic force applied to the sails, the hydrodynamic force applied to the hull and appendages and the force due to gravity [2]. For ideal steady motion, the sum of these forces and of their associated moments are both equal to zero. The sails develop a thrust and a lateral force that are respectively equal to the hydrodynamic resistance and lift generated by the hull, keel and rudder. Moreover, the aerodynamic lateral force and the hydrodynamic lift generate a heeling moment that must be compensated by the righting moment of the hull.

As a rule of the thumb, a boat is fast sailing upwind when it is stable (i.e., large and heavy and thus able to carry large sails even in strong wind) and when the sails and hull have a large efficiency (lift/drag ratio). On the contrary, when sailing downwind the boat should be light and narrow with the sails acting like parachutes to develop the maximum thrust. For the America's Cup, the race course is composed of a number of upwind and downwind legs. The fastest yacht exhibits the best compromise between upwind and downwind performances. In addition, each boat must comply with a class rule that imposes a relation between the boat length, displacement and sail area, with the goal of equalising the average speed of all the competing yachts. This rule, while representing a good average, is too simple to account in detail for the performances of different yachts in different weather conditions. The goal of each competitor is to take advantage of the simplistic nature of the rule, designing a yacht faster than that predicted by the rule. To achieve this, detailed knowledge of the forces acting on the boat is necessary. The approach used to predict the performance of a boat is to calculate (or measure) separately all the forces exerted on the hull and the sails as a function of the boat and wind speed, heel and yaw angles, and combinations of these variables. All these data are used by a computer code called a Velocity Prediction Program (VPP) to compute the final boat speed, and heel and yaw angles, that satisfy the equilibrium of the forces for a given wind speed and angle.

Hydrodynamic Forces

It is usual to consider the total drag of a boat sailing in calm water to be composed of wave, viscous and induced resistance. This convention has been adopted by naval architects for the purpose of towing tank experiments, however, it is not always easy to separate each component nor to take into account their mutual interaction. Sailing upwind in normal wind conditions, each of these three components represents approximately one third of the total resistance. Downwind, the induced resistance is negligible, the viscous resistance increases with the square of the boat speed, while the wave resistance has a much stronger dependence on boat speed (exponent of about 6 to 9). At high boat speed (strong winds), the wave resistance is predominant, while viscous resistance dominants at low boat speed (light winds).

Wave Resistance

Wave resistance is generated when the volume of the boat moves at the interface between the water and the air. The amount of wave resistance depends on the boat length and displacement, the shape of the hull and the Froude number (ratio of inertial and gravitational forces). While being essentially a potential flow phenomenon, some viscous boundary layer effects are present, especially close to the stern. The accurate numerical determination of wave resistance has always been extremely difficult for naval architects; for this reason, towing tanks are usually relied upon to determine the wave resistance of ships and yachts. From a computational point of view, the most efficient tools to solve potential flow problems are panel methods.

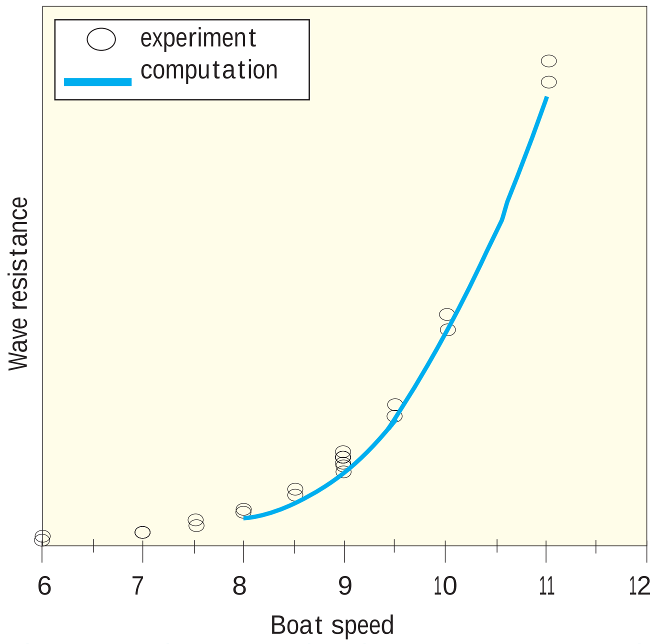

Fig.1 – Wave drag comparison between SHIPFLOW and experimental results

The commercial panel code SHIPFLOW developed by FLOWTECH International AB [3] is being intensively used at the EPFL to compute the free surface flow around yacht hulls at different speed and trim (yaw and heel angle). Rather that apply some approximative technique, SHIPFLOW uses an iterative procedure to satisfy the dynamics and boundary conditions on the exact water surface. During iteration, the trim and sink of the boat are adjusted to take properly into account the longitudinal moment generated by the towing force (applied at the centre of pressure of the sails) and the hull drag. These non-linear capabilities of SHIPFLOW are essential for the computation of the wave resistance of a sailing yacht. Indeed, one of the main problems for naval architects is to predict how the *wetted length* of the boat changes when the exact profile of the wave and the trim are taken into account. The aim is obviously to design a boat shape that at speed has the greatest wetted length for the same length calculated statically by the above-mentioned class rule.

Previous attempts to calculate wave resistance of sailing boats with linear panel codes (where the boundary conditions are satisfied at the undisturbed flat-water surface) gave disappointing results. However, using the non-linear SHIPFLOW code, a good comparison between computations and the experimental data available has been obtained, as shown in fig. 1.

In addition to the quantitative determination of the wave resistance, visualisation of the computed flow fields allows an easy comparison of the pressure distribution and the wave profiles (fig. 2). Such information is very useful for the designer as an aid to understand the behaviour of different boat shapes from a qualitative point of view.

Fig. 2 – Waw profile around an IACC hull computed by SHIPFLOW

Viscous and Induced Resistance

Despite its importance, little attention is usually paid by naval architects to predict correctly the value of the viscous resistance. In the classical towing tank methodology, the friction drag is assumed to be equal to that of a flat plate having the same wetted area as the hull. Of course, the variable pressure distribution acting on the hull results in a behaviour of the boundary layer that is different than that of a flat plate. The ratio of the actual viscous resistance to that of the equivalent flat plate is called the “form factor”, a quantity that is difficult to measure in a towing tank. It should be noted that a poorly-determined form factor can lead to an incorrectly-designed stern shape, which may give rise to undesired flow separation.

To determine the viscous resistance of an IACC yacht, three-dimensional Navier-Stokes computations for a number of different hull shapes are being performed at the EPFL using the commercial code FLUENT/UNS [4]. Each hull is usually tested at different speed and trim (fig. 3). The free surface is considered either as a flat undisturbed surface or using the perturbed free surface computed by SHIPFLOW. An accurate modelling of the boundary layer is required to calculate the skin friction and the occurrence of flow separation. A hybrid computational mesh is therefore adopted, using unstructured tetrahedral elements in the bulk flow, and structured prismatic elements in the boundary layer. Usually a total of 1.5 to 2 million cells is used, with about half of the cells located in the boundary layer. The computational time for the 500 iterations required for convergence is typically 10 hours on 12 processors of a Silicon Graphics Origin2000.

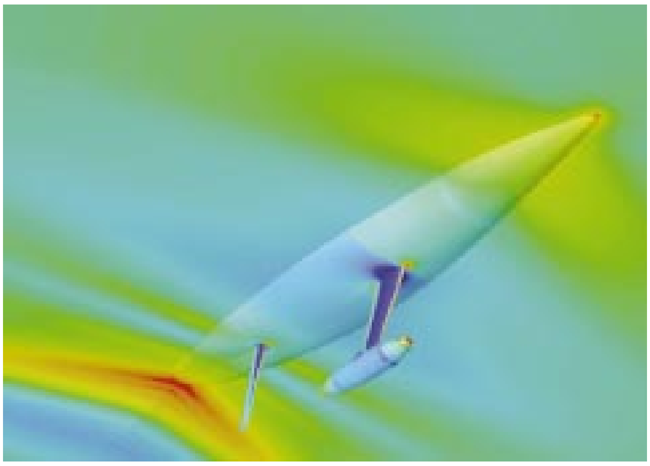

Fig. 3 – Pressure distribution around an IACC hull and appendages computed by FLUENT/UNS

When the boat sails with a yaw angle (angle of attack), the hull and the appendages develop a lift and an induced resistance that is determined by the vortex shedding. For wing-type surfaces, such as the keel and rudder, the location of the vortex separation can be easily predicted at the sharp trailing edge. A vortex smoothly detaches also on the leeward side of the hull, and in this case the location of the detachment and the strength of the vortex is strongly dependent on the pressure distribution along the hull and the behaviour of the boundary layer (see figure on cover page). Additional vortices are shed at the junction between the appendages and the hull (horseshoe vortex) that generate the so-called interference drag; this can be reduced with properly designed fillets and streaks. The strength and trajectory of all these vortices must be correctly calculated to predict the total lift and induced resistance of the boat and the optimum distribution of lift between the keel and the rudder.

Aerodynamic Flow

Sails are surfaces that must develop the maximum lift with the minimum drag when sailing upwind and the maximum drag downwind. Upwind they should work as very efficient wings with the optimal circulation distribution and the lowest flow separation. Sails are not rigid wings, but thin flexible surfaces; this has some advantages and some drawbacks. The 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 suddenly separates if the sail is not working at the ideal angle of attack.

Determination of Flying Shape of Sails

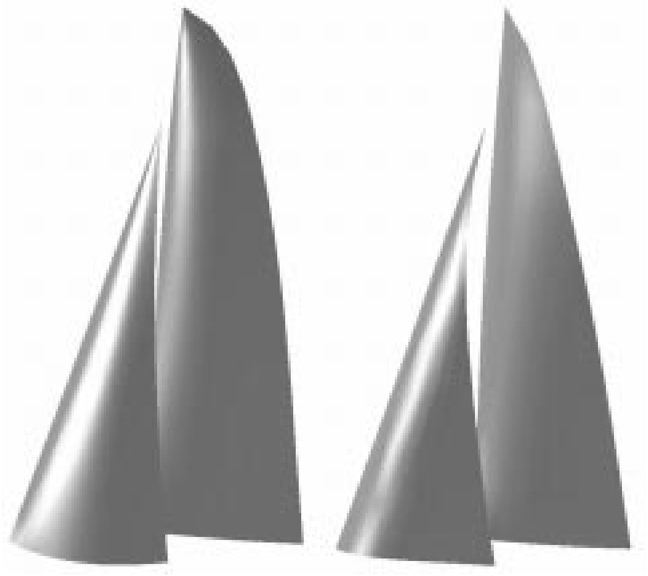

Fig.4 – Design (left) and flying (right) shapes of IACC upwind sails, the latter determined by FLOW-MEMBRAIN aeroelastic computations

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. This is performed at the EPFL using the codes FLOW and MEMBRAIN, developed by North Sails Inc. FLOW is a panel code 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. 4 shows the results of a computation, where the initial and flying shape of a sail plan can be compared. 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), yielding the best-efficiency operating point of the sail.

Upwind Sail Computations

A potential flow computation using a panel code is able to give a precise aerodynamic load distribution only if the flow is attached. Under this hypotheses, the flying shape, lift and induced resistance are satisfactory predicted. However, even with well-trimmed sails some amount of separation always exists and this affects the final efficiency of the sails. In practice, it is important to know how the optimum camber of the sails varies with the wind speed, where the optimum is the best compromise between maximum lift achievable and minimum viscous resistance. Such effects can only be determined by a viscous flow computation. This can be performed with FLUENT/UNS, using the flying shape computed by FLOW-MEMBRAIN. Numerical simulations have been performed using a flow domain represented by about 1 million cells, with about 500 iterations needed to obtain convergence. A computational time of about 8 hours is required on 12 processors of the Silicon Graphics Origin2000.

One of the principal sources of aerodynamic drag is the mast, which can considerably increase the total aerodynamic resistance of the sail. Usually the flow separates on both sides of the mast and then reattaches on the mainsail (see fig.5). Lift is not affected in general, but the efficiency decreases. From the design point of view it is important to analyse this phenomenon in order to be able to design a mast profile that minimise the flow separation. A number of mast profiles have been studied using FLUENT/UNS. Fig. 6 shows a comparison between the computed pressure distribution and the experimental results obtained in the wind tunnel on a simple circular mast [5].

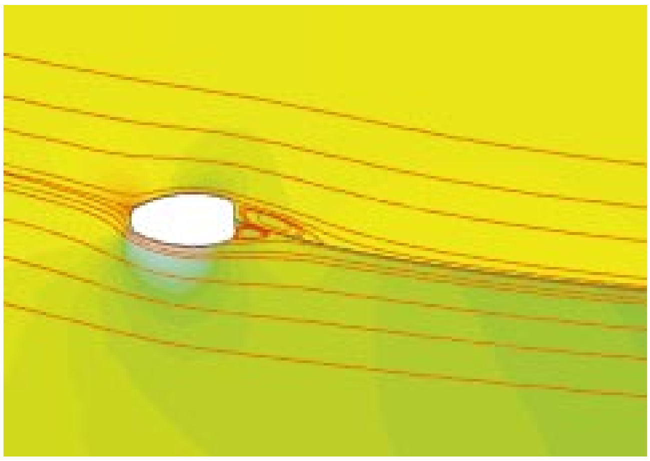

Fig. 5 – Flow around an IACC mast and mainsail geometry computed by FLUENT/UNS

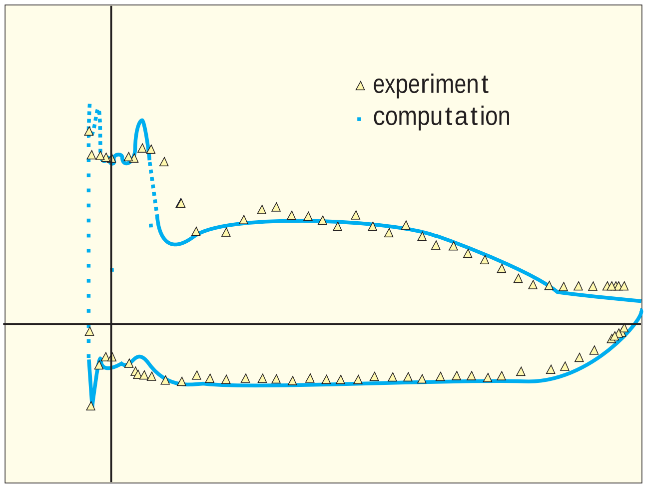

Fig. 6 – Comparison between FLUENT/UNS and experimental 2D pressure distribution around a mast and mainsail geometry

Downwind Sail Computations

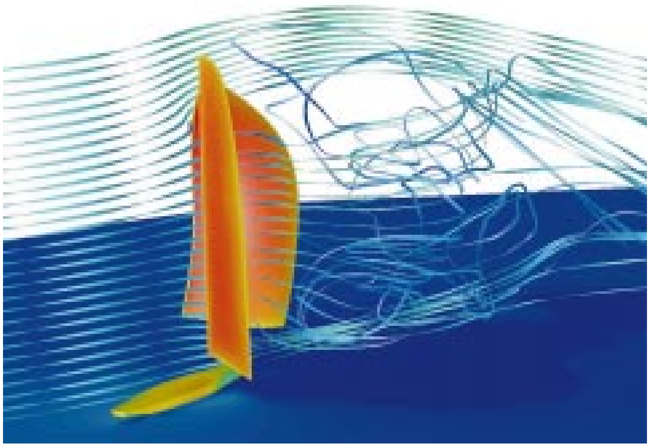

Fig. 7. – Surface pressure contours and streamlines for flow around a downwind sail configuration of an IACC yacht computed by FLUENT/UNS

While panel codes have been used with success to compute the forces acting on upwind sails, this is not possible for downwind sails due to the presence of large separated flow regions. A large potential for design improvement therefore exists in the use of advanced viscous flow codes for downwind flow configurations. Computations have been performed using FLUENT/UNS of the flow around such a downwind configuration, composed of a spinnaker and mainsail as well as the exposed section of the hull. (Such simulations required similar computational resources as for the viscous flow computations of the upwind configurations.) As shown in fig. 7, a large separated wake region is observed downwind of the spinnaker, comprised of a complex series of vortices. While it is not possible to compute accurately all the features of this extremely complicated flow behaviour, it is anticipated that such numerical simulations can provide a detailed knowledge of the interactions between the mainsail, spinnaker and hull. In addition, a study of the dependence of these interactions, as well as the general flow behaviour, for different sail configurations is of great interest for design purposes.

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 show that, despite the extremely complex flow behaviour present, information can be gained that is invaluable for design purposes. This information can be either 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 America's Cup yachts is entering a new phase through the use of state-of-the-art numerical flow solvers on high-performance parallel 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 authors wish to acknowledge the intense interest in this work by members of the FAST 2000 Design Team (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. This work has received financial support from the Commission pour la Technologie et l'Innovation.

References

[1] M.-C. Sawley, FAST2000, un projet toutes voiles dehors, EPFL Supercomputing Review, 9, 18-20 (1997).

[2] J.H. Milgram, Fluid mechanics for sailing vessel design, Annual Review of Fluid Mechanics, 30, 613-653 (1998).

[3] Shipflow User's Manual; see also www.flowtech.com

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

[5] S. Wilkinson, Ph.D Thesis, "Partially separated flow around masts and sails", University of Southampton, Ship Science Dpt., 1984. ■