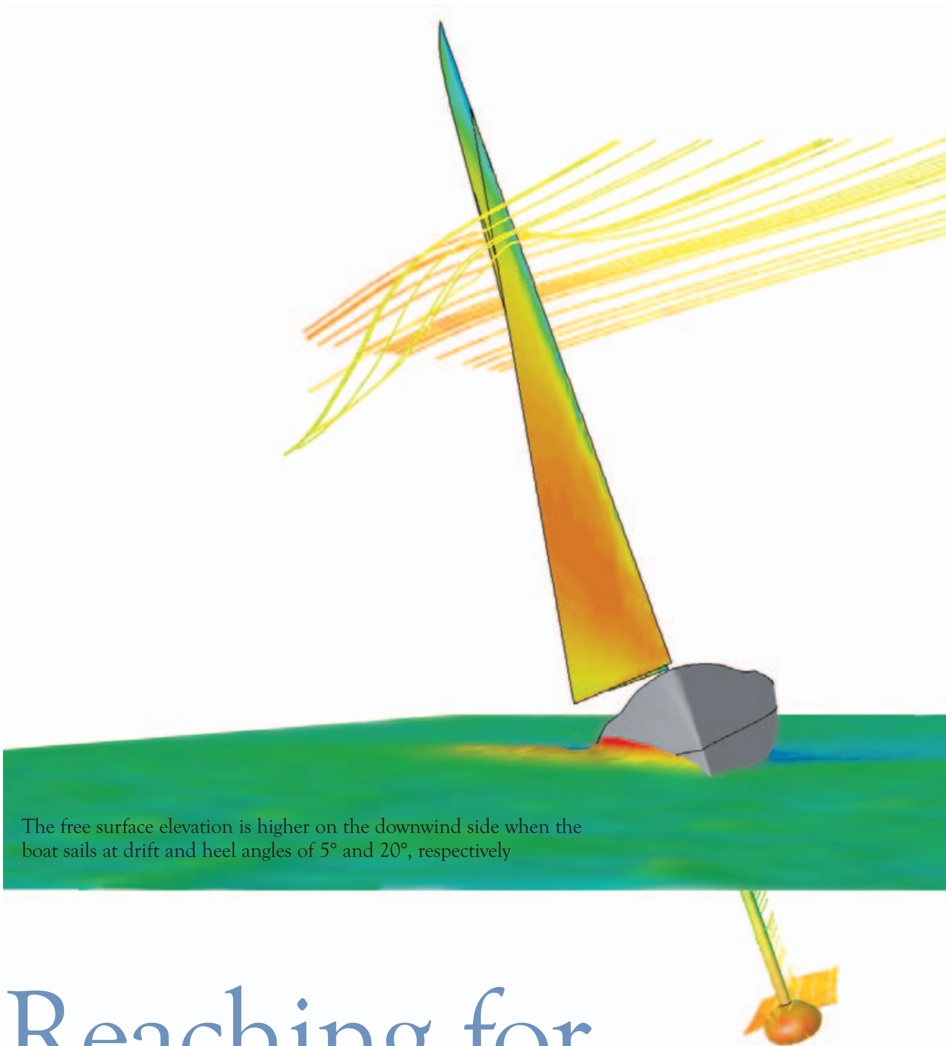
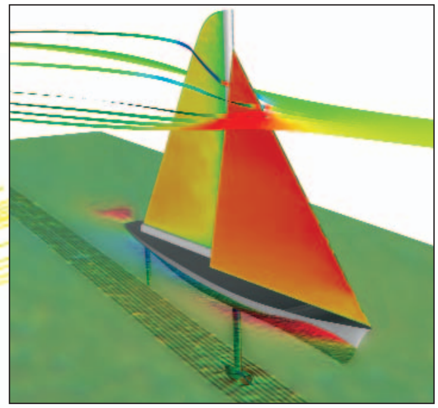
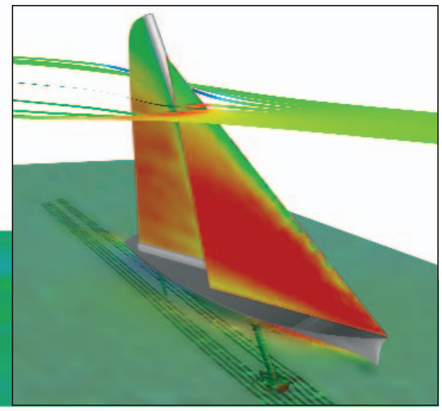


The free surface elevation is higher on the downwind side when the boat sails at drift and heel angles of 5° and 20°, respectively

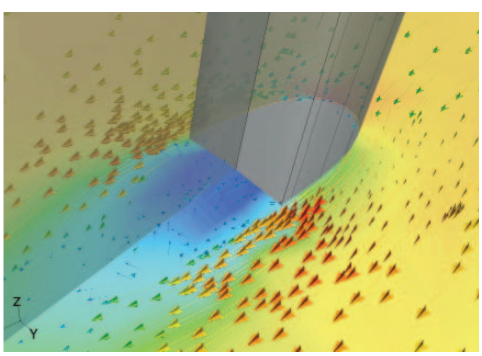


Pathlines, the water elevation, and pressure contours on the sail, keel, and bulb for drift and heel angles of 0° and 0° (top) and 5° and 20° (bottom), respectively



Reaching for Better Yacht Design

by Filippo Lupo, Davide Lucarelli, TESCO TS SpA, Torino, Italy



Airflow separation behind the mast

THE DESIGN OF MODERN SAILING YACHTS is based both on classical towing tank tests and numerical flow simulations. CFD simulations can be a useful aid for yacht designers to understand the complex physics of the flow phenomena involved in the aerodynamics of the mast and sails, the hydrodynamics of the hull and appendages, and the strong interaction between the two. CFD simulations require less time than experiments, but represent important support for experiments. Using numerical simulation, different geometries can be tested before scale models are created for towing tank or wind tunnel tests. CFD also provides detailed flow field visualization, which is difficult to obtain by experimental means.

TESCO TS is an Italian company within the PCL Group, a leading provider of engineering services for the product development process, with customers in the automotive, aerospace, and marine markets. Their offerings range from the provision of on-site engineering capacity to complex outsourced work packages at different locations, such as Germany, Italy, France, the UK, Sweden and Japan. TESCO's expertise in CFD analysis using FLUENT covers external aerodynamics, multiphase hydrodynamics, aeroacoustics, HVAC and climate comfort, engine cooling, and in-cylinder flows. Recently, TESCO engineers applied FLUENT 6.2 to the study of a complete sailing yacht.

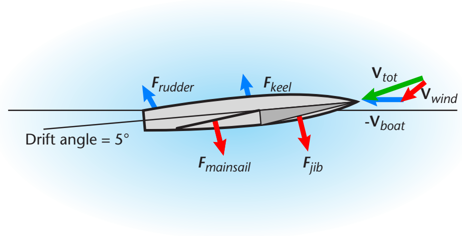
Usually, CFD studies of sailing yachts separate the aerodynamic and hydrodynamic phenomena, although studies of the deformed free surface near the waterline have been performed. While the separation of aero- and hydrodynamics saves time for model building and simulation, it doesn't take into account the strong interaction between the forces associated with each. Moreover the role of viscous effects on the free surface elevation is often neglected, and this prevents such things as water flow separation (close to the stern during upwind cruising with certain drift and heel angles, for example) from being properly captured.

The aim of the present work was to establish a methodology for performing a coupled CFD analysis of the aerodynamic flow around the mast and sails, together with the hydrodynamic flow around the hull and appendages, including viscous effects and surface wave generation at the waterline. The results of particular interest from the analysis include the free surface elevation with the related wave drag, and the aerodynamic and hydrodynamic lift and drag acting on the sails, keel, and rudder. The lift and drag results can be used to derive the thrust and lateral forces on the yacht, which can then be used to predict the boat velocity corresponding to a fixed configuration in terms of sink and trim, as well as yaw, drift and heel angles.

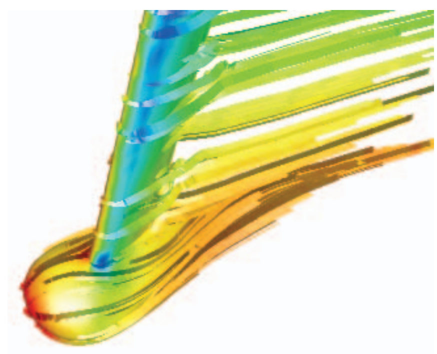
The volume of fluid (VOF) model in FLUENT was used for the analysis. This multiphase model is designed for immiscible fluids (air and water in this case), and tracks the interfaces between them. The dynamic mesh tool was also used for some of the simulations to introduce transient yacht motion (changes in drift and heel angles) into the calculation. Another goal of the analysis was to provide detailed flow visualization in certain critical areas, such as the air flow separation zone on the mainsail behind the mast, and the water flow separation zone close to the rudder near the stall angle.

An unstructured hybrid mesh of approximately 2 million cells, representing a model sailing yacht in a towing tank/wind tunnel, was created using TGrid, with a surface mesh imported from ANSA. The towing tank and wind tunnel were divided by an initially undisturbed free surface in order to apply different boundary conditions to each region (for velocity, for example) and to refine the mesh on the air/water interface. The unsteady simulations were performed using the VOF model along with the k-ε model for turbulence on four processors of a Linux cluster. It took about two days to complete a simulation time of four seconds.

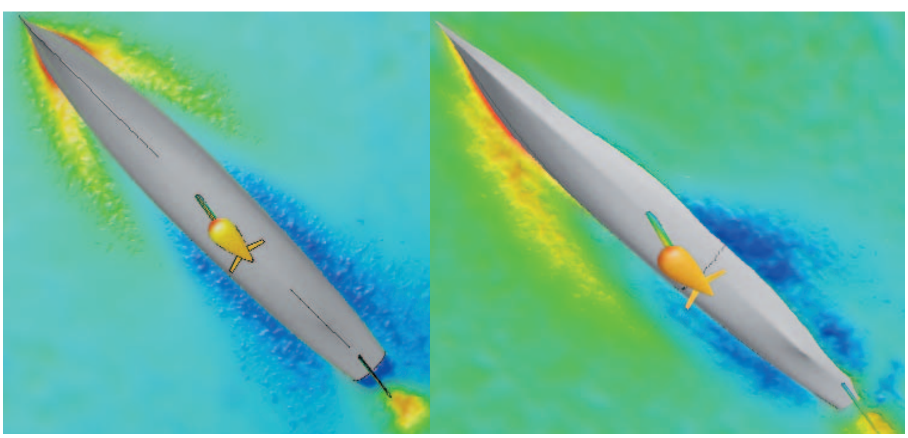
The coupled analysis of wind flow around the sails and water flow around the hull and appendages has proved to be a powerful tool for predicting yacht performance. It has provided insight into the simultaneous aerodynamic and hydrodynamic lift and drag on the sails, keel, and rudder in less time and at less cost than would be required for towing tank and wind tunnel tests. Moreover, the airflow separation on the mainsail behind the mast can be used to suggest new designs for the mast cross-section. The wave drag at the free surface can be used to derive the thrust and lateral forces, and in the future, the computational methodology will be extended to include the six-degrees-of-freedom (6DOF) solver, which will allow the boat to move as result of these forces (rather than fixed, prescribed boundary conditions). ■



A vector diagram showing the wind and boat velocities, the resultant velocity, and some of the forces on the boat; a drift angle of 5° is shown



Pathlines emanating from the bulb and keel for drift and heel angles of 5° and 20°, respectively; the surface contours and pathlines are colored by pressure coefficient



The surface elevation of the water for drift and heel angles of 0° and 0° (left) and 5° and 20° (right), respectively