Proceedings of the Eighteenth (2008) International Offshore and Polar Engineering Conference
Vancouver, BC, Canada, July 6-11, 2008

ISBN 978-1-880653-70-8 (Set); ISBN 1-880653-68-0 (Set)

Numerical Simulation of a flow around an America's Cup class Keel

M. M. Ambrogi, R. Broglia, A. Di Mascio
INSEAN – Italian Ship Model Basin
Rome Italy

ABSTRACT

In this work the flow field around an America's cup class Keel is studied by the numerical solution of the Reynolds Averaged Navier-Stokes equations (RANSE). An analysis is carried out in order to assess the efficiency of the winglet in reducing the induced resistance of the keel. Velocity, pressure and vorticity fields have been analyzed to improve the understanding of the flow field around such a peculiar hydrodynamics appendage. Velocity and pressure fields, as well as global loads for the simulations with and without winglets configurations are compared. Comparison with available experiments is made.

KEY WORDS: Winglet; vorticity; induced vortex; turbulent flows; RANS based computations.

INTRODUCTION

One of the first yacth to introducd the winglets in keel design is Australia II in 1983, but their usefulness is still under discussion. In fact, they should obtain a reduction of the induced drag of the fin, by the generation of vortices that counter balance those around the bulb. At the same time, winglets causes a drag increment which could not be balanced by the reduction of the induced drag. This can happens because the boat attitude caused by metheorogical conditions and trim and sinkage is very complex and therefore it can operate far from design conditions, causing an overall drag increment. Another advantage lies in the production of a balancing heeling moment. It has to be noticed that this condition can be considered really useful just in high wind sailing condition. In low wind condition in fact, the boat does not tend to heel, and then the additional balancing moment is not needed, and the effect of the winglets is simply a drag increment.

A number of design aspects has to be taken into account: the winglet surface (which has a direct effect on the lift, as well as on the restoring heeling moment and a negative effects in the increment of the drag), the relative bulb position, its weight and shape. These aspects directly influence the boat center of buoyancy and the equilibrium moment. Water and enviromental conditions can be also influence the performances of the appendages; water quality, climatic conditions and sea fauna, for example, can influence turbulent transition around the geometry.

Tha aim of this paper is to analyze the flow field around a keel, whose performances are computed with and without winglet. The simulation of the effects produced by the introduction of the winglets offers a first glance in a design field up to now poorly explored by CFD.

MATHEMATICAL AND NUMERICAL MODELS

The mathematical model employed in the simulations is the one described by the Reynolds Averaged Navier Stokes (RANS) equations, the turbulent viscosity being calculated by means of the one-equation model developed by Spalart and Allmaras (1994). The problem is closed by enforcing appropriate conditions at the physical and the computational boundaries. On solid walls, velocity is set to zero (whereas no condition on the pressure is required); at the inflow boundary, velocity is set to the undisturbed flow value, and the pressure is extrapolated from inside; at the outflow, the pressure is set to zero for flow without the free surface, whereas the velocity is extrapolated from inner points. Free surface effects are not taken into account in this study.

In this section only the main aspects of the numerical method are briefly recalled; the reader interested in more details is referred to (Di Mascio ***et al.*** (2001), Di Mascio ***et al.*** (2006)). As only the average steady state has to be computed, the RANS equations can be replaced by a pseudo-compressible formulation (Chorin (1967)). In the numerical scheme, these equations are approximated by a finite volume technique with pressure and velocity co-located at cell centre. Discretization of the physical domain is achieved by means of a multi-block structured grid with partial overlapping (Muscari and Di Mascio (2005)); this technique is very convenient because it simplifies the discretization of appendages and the inclusion of blocks for local refinement. Viscous terms are computed by means of a standard second order centred finite volume approximation, while for the inviscid part, a second order Essentially Non Oscillatory (ENO) scheme has been adopted (Harten et al. (1987)). Time integration of the discrete model is achieved by means of an implicit Euler scheme; the resulting discrete system of algebraic equations is solved in "delta form" (Beam and Warming (1978)). Convergence toward steady state is accelerated by local time stepping and a multi grid algorithm (Favini ***et al.*** (1996)).

PARAMETERS AND COMPUTATIONAL GRIDS

As mentioned above the physical domain has been discretized by means of multi block structured mesh. An overlapping grid technique is used, for which an in-house Chimera algorithm is employed to manage blocks data exchange. Figure 1 shows a sketch of the computational mesh around the geometry. The total number of volumes is about seven millions.