DOWNWIND SAILS AERODYNAMICS ANALYSIS

Fabio Fossati†, Ignazio Maria Viola∗

†Politecnico di Milano CIRIVE Wind Tunnel
Politecnico di Milano, Campus Bovisa, Via La Masa 34, 20156 Milano, Italy
e-mail: fabio.fossati@polimi.it

∗Dipartimento di Ingegneria Meccanica
Politecnico di Milano, Campus Bovisa, Via La Masa 1, 20156 Milano, Italy
e-mail: ignazio.viola@tiscali.it,

Keywords: Downwind Sails, Sailing Yacht Aerodynamics, CFD, Wind Tunnel, Benchmark.

1 SHORT ABSTRACT

This paper presents some results concerning numerical and experimental research activities carried out to investigate aerodynamics of downwind sail plan configurations of IACC V5 yacht. In particular, RANS computational analyses performed using the commercial code Fluent (Ansys, Inc.) are presented and compared with experimental measures available from wind tunnel tests carried out at the Politecnico di Milano Twisted Flow Wind Tunnel. Sails flying shapes related to actual sails trimming have been detected during the wind tunnel tests and used as numerical simulation input.

2 INTRODUCTION

In recent years computational fluid dynamics has demonstrated the ability to predict sail forces in upwind conditions and in particular the recent improvements in computer performance have made the use of RANS equations feasible for practical design applications also for downwind conditions. The present work is concerned with numerical and experimental research activities carried out to investigate aerodynamics of downwind sail plan configurations and in particular in order to gain further understanding of five different sail configurations tested between close reaching and running coarse (45°, 105° and 120° apparent wind angles). At closer yaw angles, the flow is mainly attached therefore non viscous codes can well predict aerodynamic forces [1]. At larger angles, the flow is mainly separated on sail perimeters, hence global forces are less responsive to computation parameters. On the opposite, in the yaw angle range investigated, both separated and attached flow region have a strong influence in force production, lift is about twice drag force, hence none of them might be negligible. The flow around the spinnaker presents a turbulent separation bubble along the luff (i.e. leading edge), where a secondary laminar bubble, computationally un-visible, plays an important role in the first bubble dimension [2]. The reattached flow presents an accelerated boundary layer velocity profiles that is hard to catch with a standard computational wall function. Fi-
nally there is a leach (i.e. trailing edge) separation strongly connected to the tip vortex generated at the skirt (i.e. root) and at the head (i.e. tip) of the sail. The flow around the mainsail might be fully attached or separated. Moreover, at higher apparent wind angles, spinnaker is affected by alteration in the shape due to vortex shedding and consequently the separation points move along the chord. The studied phenomena presents most of the current challenge for computational fluid dynamics: transition, reattachment, re-laminarization, both laminar and turbulent separations.

3 EXPERIMENTAL TESTS

Experimental test has been performed in the Politecnico di Milano Twisted Flow Wind Tunnel [3]. Figure 1 shows the 1:12 scaled IACC model during the test: sails trimming is operated by means of 7 radio controlled winches and sailplan forces are measured by means of a 6 component force balance located inside the hull. During experimental tests, actual sail’s flying shape are detected using infrared (ir) cameras and in house developed software. The resulting sail’s flying shapes are then used do build up the CFD analyses.

4 NUMERICAL ANALYSIS

Commercial code Fluent has been used to solve the equations of the flow around the boat without considering volume forces (i.e. gravity) and density variations, therefore energy equation hasn’t been solved. SIMPLE scheme has been solved and both first and second discretization orders have been adopted. PISO scheme has also been tested but none significant effect has been notice. In the present paper, differences between laminar flow, kw standard (Wilcox98), kw-sst and Spalart-Allmaras turbulence model are pointed out. Moreover, a mesh dependency investigation has been performed considering mesh with the same topology but different wall-element-size hence difference overall cell number.

5 RESULTS

A mainsail with two different asymmetric sails (named respectively (A1) for lighter air and (A2) for stronger breeze) have been tested at 45°, 105° and 120° apparent wind angles and an additional configuration with a staysail (SS) and a different mainsail has been tested at the closer angle.

Wind tunnel tests have been performed with target velocity and twisted profiles according specific situation of an IACC yacht sailing in Valencia atmospheric boundary layer. The inlet flow boundary conditions of CFD simulations were set to give a twisted velocity profile similar to the Twisted Flow Wind Tunnel conditions.

Figure 2 shows a comparison between numerical results and experimental measurements in terms of aerodynamic thrust and side force coefficients and lift and drag coefficients (in the horizontal plane). Numerical results show good agreement with experimental data. Figure 3 shows numerical results in terms of velocity vectors plotted in a plane perpendicular to the mast (placed at mid mast height), at 45° and 120° apparent wind angle respectively. As can be seen, asymmetric sails reveal mainly attached flow while the flow on mainsail leeward is attached at 45° and separated at 120° yaw angle. Figure 4 shows Cp pressure coefficients for each sail evaluated in the same mid mast plane. Increasing the apparent wind angle, the asymmetric centre of pressure moves aft and the Cp curve shapes show attached flow for more than 70% of the chord length for both yaw angles. On the contrary, mainsail curve shapes reveal a fully attached and fully separated flow.
In Figure 5 numerical results computed with Spalart-Allmaras turbulence model (sa), kw standard and kw sst turbulence models are compared to numerical results computed with laminar flow (lam) equations, both in terms of aerodynamic coefficients and centre of effort height ZCE. In Figure 5 linear trend illustrate that turbulence model affects mainly drag, and hence side component force, then lift. Furthermore the influence increases with the model complexity. Note that laminar simulation always presents the better agreement with the experimental data. In Figure 6 computational results for different mesh size from 1 million cells to 36 millions cells are compared to experimental data. In the same figure drag and lift results trends are fitted by a second order polynomial curves. All meshes are built with tetrahedral cells, with the same grow rate, increasing from sails to external domain boundaries, and with the same maximum cell dimension. Meshes differ for wall-region cell dimension; this means that mesh size affects both boundary region and wake region.
6 CONCLUSION

Five downwind configurations tested at Politecnico di Milano Twisted Flow Wind Tunnel have been processed with RANS techniques. Computational parameters as mesh size, turbulence models, etc, have shown great influence in forces prediction. Numerical investigations demonstrate that a suitable parameter setting leads to properly predict sailplan forces with reasonable computational efforts. Numerical accuracy let to compare different sails, trims, and achieve strategic decision as the opportunity to sail with or without staysail. Furthermore, numerical flow visualization allows a better understanding in downwind aerodynamics.

ACKNOWLEDGEMENTS

The authors would like to acknowledge the 32th America’s Cup LunaRossa Challenger 2007 for funding research activities and PhD grant.

7 REFERENCES

