Computational Fluid Dynamics of Sail Flow

Jonathan Paton and Hervé P. Morvan

CFD Group, School of Civil Engineering,
The University of Nottingham, Nottingham, NG7 2RD, UK
Herve.Morvan@Nottingham.ac.uk

Interest in optimum sail design has led to the increasing use of Computational Fluid Dynamics technology. Collie [1] in particular focuses on the detailed computation of the effects of downwind sailing on high camber sails such as spinnakers. The key issue associated with downwind sailing is flow separation. This paper replicates and expands one of Collie's test cases; high level discretisation is employed as well as a recent turbulence models, namely SST and SAS [2]. The aim of this exercise is for the authors to assess the accuracy of the CFD results for drag and lift for a sail-like geometry ahead of a detailed fluid-structure interaction (FSI) investigation of a sail. Noteworthy recent contributions to FSI of sails are the works of Shankaran [3] and Mairs [4]; however both relied on simplistic fluid models, either ignoring viscous and turbulent terms [3] or relying on a simple turbulence model with poor aerodynamics credentials [4]. The authors' work should remediate both shortcomings.

A circular arc of chord length 319 mm rotated every 2.5 degrees between 5 and 30 degrees is investigated in 2D for a mean flow of 25 m/s. Validation results for the test case and 2-D numerical results are available in [1]. At the higher angles of attack Collie [1] reports a continuing increase in lift believed to be caused by the high blockage ratio and wall effects of the small wind tunnel. However Collie [1] chooses to use the same dimensions for the wind tunnel and computational domain, and replicates the phenomenon. In the present study the domain blockage ratio is significantly reduced to alleviate this issue by way of an enlarged computational domain and the authors are finding that the lift coefficient reaches a plateau at high angle of attacks, which does not match the data, but seems more realistic. Furthermore whilst Collie [1] obtains a smooth rise in the lift coefficient with increasing angles of attack the authors are able to capture the collapse of the lift coefficient in the region 10-20 degrees evidenced in the experimental data. The computation of this effect is key to the next phase of the investigation proposed by the authors when they will be coupling their CFD work to a membrane model for the sail. The authors' results for the drag coefficient are over predicted for the high angle of attack, although less than in earlier works.

Promising results are also presented for the same test case with the SAS model in 2-D, and subsequently in 3-D to assess the turbulent and 3-D nature of the flow for such downwind sailing conditions.

References:

[1] S. Collie, "Application of Computational Fluid Dynamics to Two-Dimensional Downwind Sail Flows", *PhD Thesis*, The University of Auckland, New Zealand, 2005.

[2] F. Menter and Y. Egorov, "a Scale-Adaptive Model using Two-Equation Model", *AIAA* paper 2005-1095, 2005.

[3] S. Shankaran, "Numerical Analysis and Design of Upwind Sails", *PhD Thesis*, Stanford University, USA, 2005.

[4] Mairs, "Aerodynamic-Structural Model of Off-Wind Yacht Sails", *PhD Thesis*, The University of Maryland, USA, 2003.

Keywords: CFD, Downwind, Sailing, Fluid, Structure, Interaction.